

**AFRL-VA-WP-TR-2005-3120**

**HYPERSONIC ENGINEERING  
AEROTHERMODYNAMIC  
TRAJECTORY TOOL KIT (HEAT-TK)  
Delivery Order 0009: Software User's Manual**



**Michael L. Carter, Geojoe Kuruvila, Yuk Woo, Kei Y. Lau, and Kevin G. Bowcutt**

**The Boeing Company  
Phantom Works  
P.O. Box 3999  
Chicago, IL 60606**

**JANUARY 2005**

**Final Report for 11 July 2001 – 01 January 2005**

**Approved for public release; distribution is unlimited.**

**STINFO COPY**

**AIR VEHICLES DIRECTORATE  
AIR FORCE MATERIEL COMMAND  
AIR FORCE RESEARCH LABORATORY  
WRIGHT-PATTERSON AIR FORCE BASE, OH 45433-7542**

## NOTICE AND SIGNATURE PAGE

Using Government drawings, specifications, or other data included in this document for any purpose other than Government procurement does not in any way obligate the U.S. Government. The fact that the Government formulated or supplied the drawings, specifications, or other data does not license the holder or any other person or corporation; or convey any rights or permission to manufacture, use, or sell any patented invention that may relate to them.

This report was cleared for public release by the Air Force Research Laboratory Wright Site (AFRL/WS) Public Affairs Office and is available to the general public, including foreign nationals.

Copies may be obtained from the Defense Technical Information Center (DTIC) (<http://www.dtic.mil>).

AFRL-VA-WP-TR-2005-3120 HAS BEEN REVIEWED AND IS APPROVED FOR PUBLICATION IN ACCORDANCE WITH ASSIGNED DISTRIBUTION STATEMENT.

//Signature//

---

DANIEL TEJTEL  
Aerospace Engineer  
Aerodynamic Configuration Branch  
Aeronautical Sciences Division

//Signature//

---

JAMES H. MILLER, Ph.D.  
Chief  
Aerodynamic Configuration Branch  
Aeronautical Sciences Division

//Signature//

---

MICHAEL J. STANEK, Ph.D.  
Technical Advisor  
Aeronautical Sciences Division

This report is published in the interest of scientific and technical information exchange, and its publication does not constitute the Government's approval or disapproval of its ideas or findings.

REPORT DOCUMENTATION PAGE				Form Approved OMB No. 0704-0188	
<p>The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number. <b>PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.</b></p>					
1. REPORT DATE (DD-MM-YY) January 2005		2. REPORT TYPE Final		3. DATES COVERED (From - To) 07/11/2001– 01/01/2005	
4. TITLE AND SUBTITLE HYPERSONIC ENGINEERING AEROTHERMODYNAMIC TRAJECTORY TOOL KIT (HEAT-TK) Delivery Order 0009: Software User's Manual				5a. CONTRACT NUMBER F33615-00-D-3052-0009	
				5b. GRANT NUMBER	
				5c. PROGRAM ELEMENT NUMBER 0602201	
6. AUTHOR(S) Michael L. Carter, Geojoe Kuruvila, Yuk Woo, Kei Y. Lau, and Kevin G. Bowcutt				5d. PROJECT NUMBER A06C	
				5e. TASK NUMBER	
				5f. WORK UNIT NUMBER 0A	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)  The Boeing Company Phantom Works P.O. Box 3999 Chicago, IL 60606				8. PERFORMING ORGANIZATION REPORT NUMBER	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)  Air Vehicles Directorate Air Force Research Laboratory Air Force Materiel Command Wright-Patterson Air Force Base, OH 45433-7542				10. SPONSORING/MONITORING AGENCY ACRONYM(S) AFRL-VA-WP	
				11. SPONSORING/MONITORING AGENCY REPORT NUMBER(S) AFRL-VA-WP-TR-2005-3120	
12. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release; distribution is unlimited.					
13. SUPPLEMENTARY NOTES Report contains color. PAO Case Number: AFRL/WS 06-2072, 25 Aug 2006.					
14. ABSTRACT  The objective of this effort is to modify an existing engineering code to provide fast prediction capabilities to assess the aerothermodynamic performance characteristics of advanced high speed configurations throughout a flight trajectory. The prediction techniques are to be engineering methods suitable for preliminary design analysis and configuration tradeoff studies. The code expands the core capability of the current Viscous Effects on Complex Configuration (VECC) which was originally developed for the Air Force by Boeing's team in St. Louis. The final aerothermodynamic trajectory analysis tool kit, named Hypersonic Engineering Aerothermodynamic Trajectory Tool Kit (HEAT-TK), incorporate modern geometry handling capabilities, a modern GUI, updated engineering aerodynamic heating analysis methods, modern data handling (including material thermal properties) and integration methods, three degree-of-freedom trajectory analysis, aeroheating/thermal analysis along trajectories, and a two-dimensional/three-dimensional analysis output display capability with real-time viewing manipulation. All coding are performed with a modular, open-architecture philosophy.					
15. SUBJECT TERMS aerodynamics, aerothermodynamics, high speed, hypersonics, trajectory method, streamlines, aeroheating					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT: SAR	18. NUMBER OF PAGES 96	19a. NAME OF RESPONSIBLE PERSON (Monitor) Daniel Tejtcl  19b. TELEPHONE NUMBER (Include Area Code) N/A
a. REPORT Unclassified	b. ABSTRACT Unclassified	c. THIS PAGE Unclassified			

## Table of Contents

Section	Page
List of Figures .....	v
List of Tables .....	vii
1 Overview .....	1
1.1 The HEAT-TK System .....	1
1.2 The HEAT-TK Graphical User Interface .....	4
1.2.1 Main Window .....	4
1.2.2 Workflow Window .....	8
1.2.3 Two-D Plot Window .....	10
1.2.4 Three-D VRML Viewer .....	11
1.2.5 HEAT-TK Parameter Settings .....	15
2 Geometry Modeling .....	18
2.1 IGES Files .....	18
2.1.1 Additional IGES Restrictions Imposed by HEAT-TK .....	18
2.1.2 Description of IGES Entities Used by HEAT-TK .....	19
2.2 Set Panel Attributes .....	23
2.3 Grouping panels .....	24
3 Streamline Analysis (Quadstream) .....	26
3.1 Load Files .....	26
3.2 Set Panel Types .....	27
3.3 Group Panels .....	27
3.4 Specify Parts .....	27
3.5 Set Controls .....	28
3.6 Set Start/Stop .....	29
3.7 Save Inputs .....	30
3.8 Run Quadstream .....	31
3.9 View Streamlines .....	32
4 High Speed Aero Analysis (S/HABP) .....	34
4.1 Load Files .....	34
4.2 Set Panel Types .....	35
4.3 Group Panels .....	35
4.4 Set Up Trajectory .....	35
4.5 Define Analysis Runs .....	37
4.6 Select Runs .....	38
4.7 Specify Trim .....	39
4.8 Save Inputs .....	40
4.9 Run S/HABP .....	41
4.10 Generate Aero Table .....	42
4.11 Run Trim .....	43
4.12 View Aero Results .....	44
4.12.1 Two-D Plots .....	44
4.12.2 3D Surface Map .....	45
5 Low Speed Aero Plug-in .....	47
5.1 Setup Script .....	47

5.2	Run Script .....	48
6	Trajectory Plug-in .....	50
6.1	Generate Otis Data.....	50
6.2	Setup Script.....	51
6.3	Run Script .....	52
6.4	Aero Table Format.....	53
7	Aeroheating Analysis (S/HABP) .....	54
7.1	Load Files.....	54
7.2	Set Panel Types.....	55
7.3	Group Panels.....	55
7.4	Set Up Trajectory.....	56
7.5	Set Up Component Analysis.....	59
7.6	Save Inputs.....	61
7.7	Run Program .....	62
7.8	View Aeroheating Results .....	63
7.8.1	Two-D Element Plots.....	63
7.8.2	3D Surface Map .....	64
7.8.3	Write Tecplot .....	65
8	TPS Sizing Analysis (Chap) .....	67
8.1	Load Files.....	67
8.2	Run Model Selection.....	68
8.3	Specify Elements .....	71
8.4	Run Pre-Sizing.....	72
8.5	Run TPS Sizing.....	73
8.6	TPS Sizing Programs .....	74
8.6.1	Selecttps1 .....	74
8.6.2	Prechap2.....	76
8.6.3	Chap .....	76
9	References.....	77
Appendix A	Subsonic – Supersonic AEROdynamics (SSAERO) .....	78
	Geometry Discretization .....	78
	Aerodynamic Analysis Methods.....	80
Appendix B	Mission Analysis Program (MAP) .....	82

# List of Figures

Title	Page
Figure 1 HEAT-TK System Components.....	1
Figure 2 Main Window - Geometry Analysis.....	5
Figure 3 Main Window - Aero Analysis.....	6
Figure 4 Main Window - Trajectory Generator.....	6
Figure 5 Main Window - Aeroheat/TPS Sizing.....	7
Figure 6 Main Window - Tool Set.....	8
Figure 7 Sample Workflow Window.....	9
Figure 8 Sample 2D Plot Window.....	11
Figure 9 Plot Controls Window.....	11
Figure 10 Sample 3D Controls Window.....	12
Figure 11 Sample 3D Viewer Window.....	15
Figure 12 HEAT-TK Parameter Settings.....	16
Figure 13 Type 143 Entity in Physical and Parametric Space.....	21
Figure 14 Type 128 entity in Physical and Parametric Space.....	22
Figure 15 Sample "Set Panel Types" Window.....	24
Figure 16 Sample "Group Panels" Window.....	25
Figure 17 Streamlines - Load Files.....	26
Figure 18 Streamlines - Specify Parts.....	28
Figure 19 Streamlines - Set Controls.....	29
Figure 20 Streamlines - Set Start/Stop.....	30
Figure 21 Streamlines - Save Inputs.....	31
Figure 22 View Streamlines.....	33
Figure 23 High-Speed Aero - Load Files.....	35
Figure 24 High-Speed Aero - Set Up Cases.....	36
Figure 25 High-Speed Aero - Define Analysis Runs.....	38
Figure 26 High-Speed Aero - Select Runs.....	39
Figure 27 High-Speed Aero - Specify Trim.....	40
Figure 28 High-Speed Aero - Save Inputs.....	41
Figure 29 High-Speed Aero - Run S/HABP.....	42
Figure 30 High-speed Aero - Generate Aero Table.....	43
Figure 31 High-speed Aero - Run Trim.....	44
Figure 32 High-Speed Aero - 2D Plots.....	45
Figure 33 High-Speed Aero - 3D Surface Map.....	46
Figure 34 Low-Speed Setup Script.....	47
Figure 35 Low-Speed Run Script.....	49
Figure 36 Generate Otis Data for Trajectory Analysis.....	50
Figure 37 Trajectory Setup Script.....	51
Figure 38 Trajectory Run Script.....	52
Figure 39 Trajectory Aeroheating - Load Files.....	55
Figure 40 Trajectory Aeroheating - Set Up Trajectory.....	56
Figure 41 Example of Type 10 Trajectory Data File.....	57
Figure 42 Example of Type 70 Trajectory Data.....	58

Figure 43 Example of Type 80 Trajectory Data File .....	58
Figure 44 Trajectory Aeroheating - Set Up Component Analysis.....	60
Figure 45 Trajectory Aeroheating - Save Inputs.....	62
Figure 46 Trajectory Aeroheating - Run Program.....	63
Figure 47 Trajectory Aeroheating - 2D Plots .....	64
Figure 48 Trajectory Aeroheating - 3D Surface Map.....	65
Figure 49 Trajectory Aeroheating - Write Tecplot.....	66
Figure 50 TPS Sizing Load Files .....	67
Figure 51 A Simple TPS Model Generated for the CRI TPS .....	70
Figure 52 TPS Sizing - Run Model Selection.....	71
Figure 53 TPS Sizing - Specify Elements.....	72
Figure 54 TPS Sizing - Run Pre-Sizing .....	73
Figure 55 Run TPS Sizing .....	74

Tecplot is a registered trademark of Tecplot, Inc., Bellevue, WA, USA

Cosmo is a trademark of Computer Associates International, Inc., Islandia, NY, USA

## List of Tables

<b>Table</b>	<b>Page</b>
Table 1 HEAT-TK Analysis Codes .....	2
Table 2 HEAT-TK File Usage.....	3
Table 3 Ordering of Points in Geometry File .....	78
Table 4 Keywords to Be Used to Name Parts.....	79
Table 5 Flags to Make Various Segments of a Part.....	79
Table 6 Methods Used for Aerodynamic Analysis of the Body .....	80
Table 7 Methods Used for Aerodynamic Analysis of Lifting Surfaces.....	81



# 1 Overview

The Hypersonic Engineering Aerothermodynamic Trajectory Tool Kit (HEAT-TK) provides an environment for doing aerothermodynamic trajectory analysis. It is an upgrade and extension to the Viscous Effects on Complex Configurations (VECC) analysis system, originally developed for the Air Force by The Boeing Company. HEAT-TK is suitable for performing preliminary design analysis and tradeoff studies of advanced high-speed aerospace vehicle configurations throughout a flight trajectory. It is also capable of predicting surface temperatures and heat fluxes, plus sizing of Thermal Protection Systems (TPS), on the surfaces of vehicle configurations. This document is a User's Manual for HEAT-TK, focusing on the HEAT-TK Graphical User Interface (GUI). The remainder of Section 1 will discuss the overall architecture of HEAT-TK and how the GUI is organized.

## 1.1 The HEAT-TK System

The HEAT-TK system consists of several modules as shown in Figure 1. The primary module is the Graphical User Interface (GUI), which is used to:

- ❑ guide a user through an analysis process
- ❑ view and edit geometry
- ❑ generate and view streamlines
- ❑ setup, perform, and view results for high-speed aero analysis
- ❑ setup, perform, and view results for trajectory aeroheating analysis
- ❑ setup, perform, and view results for TPS sizing
- ❑ incorporate low-speed aero data

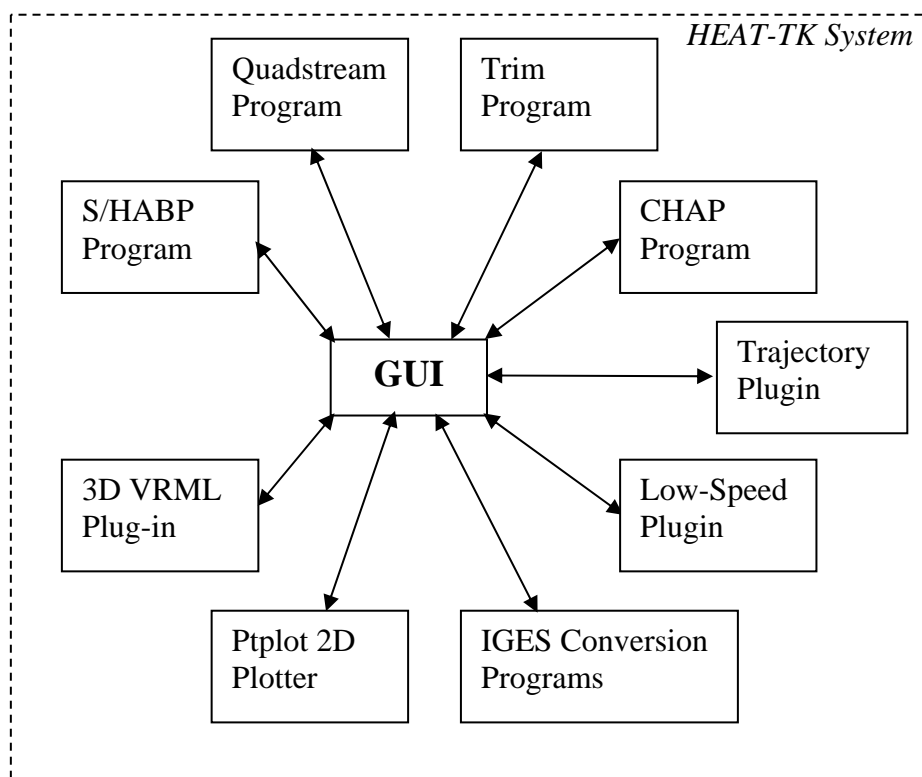


Figure 1 HEAT-TK System Components

Several analysis codes and support packages are incorporated into HEAT-TK. These analysis codes are normally (and automatically) run from within the GUI, but they can also be accessed and run separately. The following table outlines the included analysis codes and where to find associated documentation.

**Table 1 HEAT-TK Analysis Codes**

<b>Analysis Code</b>	<b>Purpose</b>	<b>Documentation</b>
Quadstream	Generate streamlines on vehicle	VECC User's Manual, Section 3
S/HABP	High-speed aero analysis	VECC User's Manual, Section 4
S/HABP	Trajectory aeroheating analysis	HEAT-TK User's manual, Section 7 (New Coding)
Trim	Compute trimmed aerodynamics	VECC User's Manual, Section 5
CHAP	TPS sizing	CHAP User's Manual
Selecttps1	TPS sizing preprocessor	HEAT-TK User's manual, Section 8.6.1
Prechap2	TPS sizing preprocessor	HEAT-TK User's manual, Section 8.6.2
SSAERO	Low-speed aero analysis	HEAT-TK User's manual, Appendix A
MAP	Trajectory analysis	HEAT-TK User's manual, Appendix B
Ptplot	2D plotting	<a href="http://ptolemy.eecs.berkeley.edu/java/ptplot/">http://ptolemy.eecs.berkeley.edu/java/ptplot/</a>
Cosmo Player plug-in	3D VRML viewer	<a href="http://www.ca.com/cosmo/">http://www.ca.com/cosmo/</a>

The HEAT-TK system relies on an extension of the standard S/HABP Mark V input file (.mk5) for storing much of its data, particularly related to streamlines, high-speed aero, trim, and trajectory aeroheating. The geometry is stored in the GEO format, although in HEAT-TK a limited capability of using Initial Graphics Exchange Specification (IGES) geometries is provided – see Section 2.1 for additional information. In using the GUI, there is seldom a need to know details about files accessed and created by the HEAT-TK system. However, for completeness, the files used by the system are given in the following table.

**Table 2 HEAT-TK File Usage**

<b>File</b>	<b>Description</b>	<b>Streamlines</b>	<b>High-Speed Aero</b>	<b>Trim</b>	<b>TPS Sizing</b>	<b>Low-Speed Plugin</b>	<b>Trajectory Plugin</b>	<b>IGES Conversion</b>	<b>Cosmo Viewer</b>
<i>file.mk5</i>	S/HABP Mark V input file	in	in		in				
<i>file.out</i>	S/HABP Mark V output file		out		in				
<i>file.geo</i>	Geometry file, geo format							out	
<i>file.iges</i>	Geometry file, IGES format							in	
<i>file.wrl</i>	VRML file								in
<i>file.plt</i>	Tecplot file		out						
<i>file.qstr</i>	Quadstream streamline file	out	in						
<i>file.qmsg</i>	Quadstream message file	out							
<i>file.qvu.N</i>	Streamline traces, VUAERO format.	out							
<i>file.qhis</i>	Streamline history file	out							
<i>file.cp</i>	Inviscid pressure coefficient data, VUAERO format		out						
<i>file.atb</i>	Aero table file		out			out	in		
<i>file.itr</i>	Trim aero input file			in					
<i>file.trm</i>	Trim aero output file			out					
<i>file.ota</i>	Otis aero table file						out		
<i>file.tml</i>	TPS sizing model file				in				
<i>file.tpl</i>	TPS sizing control data file, for selecttps1				in				
<i>file.tsb</i>	TPS sizing subtitle data file, for selecttps1				in				
<i>file.htb</i>	TPS sizing heating table file				out				
<i>file.err</i>	TPS sizing messages file, from selecttps1				out				
<i>file.psz</i>	TPS sizing elements file				out				
<i>file.pem</i>	TPS sizing panel / element / model list file				in				

<b>File</b>	<b>Description</b>	<b>Streamlines</b>	<b>High-Speed Aero</b>	<b>Trim</b>	<b>TPS Sizing</b>	<b>Low-Speed Plugin</b>	<b>Trajectory Plugin</b>	<b>IGES Conversion</b>	<b>Cosmo Viewer</b>
<i>file.szi</i>	TPS sizing CHAP deck, from prechap2				out				
<i>file.szo</i>	TPS sizing final results, summary				out				
<i>file.szf</i>	TPS sizing final results, full				out				
SHIELD.DAT. <i>n</i>	VUAERO neutral file containing shielded elements		out						
VISCOUS.ERF. <i>n</i>	Element results file for input to VUAERO		out						
VISCOUS.TP3. <i>n</i>	Type 3 geometry cards corresponding to level 2 viscous analysis		out						
POSTSC.DAT	Stability and control postprocessor aerodynamic input file		out						

## 1.2 The HEAT-TK Graphical User Interface

This section discusses the GUI design used for the HEAT-TK system. It is assumed that the user has some familiarity with using a GUI and its inherent menus and buttons. The HEAT-TK GUI mainly consists of 4 types of windows:

- ❑ The **main window** – the initial window displayed by HEAT-TK. The primary function of this window is to initiate workflow windows for selected aerothermal analyses.
- ❑ A **workflow window** that encapsulates a sequence of steps related to a specific aerothermal analysis process (e.g., High-speed aero, Streamlines, TPS sizing, ...). The sequence of steps typically includes loading file(s), setting input values, running an analysis, and viewing results.
- ❑ A **2D plot window** for displaying results from performing an analysis.
- ❑ A **3D VRML viewer** for viewing a vehicle and associated analysis results. The VRML viewer uses a web browser and a Cosmo player plug-in.

An overview of each of these windows is given in the following subsections. Some HEAT-TK parameter settings are then documented in Section 1.2.5.

### 1.2.1 Main Window

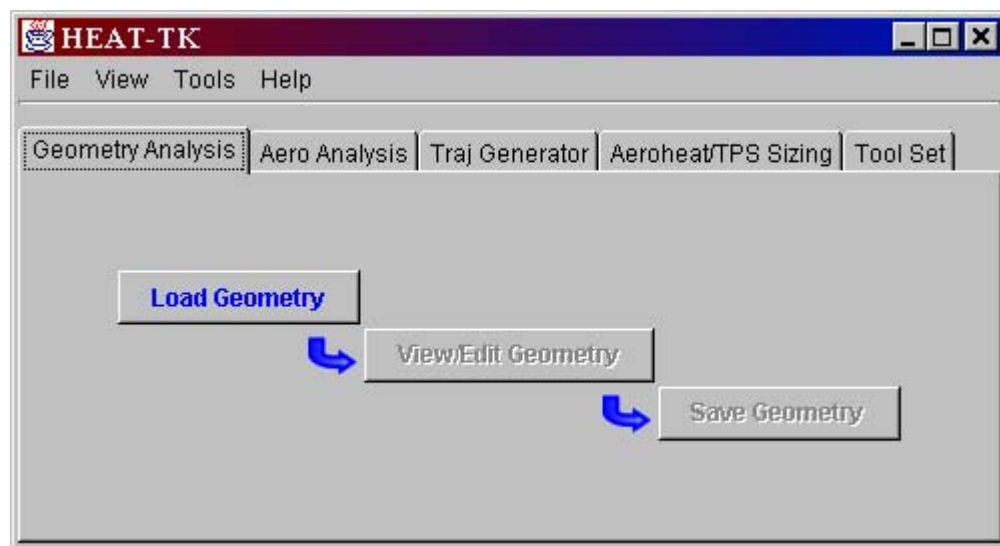
The main window is the starting point for using HEAT-TK. It consists of menu items at the top of the window and 5 tabbed panes that fill the rest of the window. Using the menu commands or the buttons on the tabbed panes, a variety of aerothermal analysis functions can be initiated. Note that all of the functions provided as menu items are also represented as buttons in the various

tabbed panes, except for the Help menu items and the Tool Options. In addition, because some analysis functions are part of more than one process, the same buttons appear in more than one of the tabbed panes. The remainder of this section provides an overview of the buttons on the 5 tabbed panes.

*Tip:* For any of the buttons on the tabbed panes, holding the mouse pointer over a button causes a “tool tip” to appear. This provides a one-line description of the button’s function.

Figure 2 shows the tabbed pane for “Geometry Analysis”. It consists of buttons that provide the following functions:

- ❑ **Load Geometry** – loads a GEO- or IGES-formatted geometry file for viewing. An S/HABP “.mk5” file that contains a GEO-formatted geometry file can also be loaded. Note that there are restrictions on the acceptable forms for an IGES file – see Section 2.1.
- ❑ **View/Edit Geometry** – brings up a window from which the geometry can be viewed in a 3D VRML viewer. This button is not activated until a geometry file has been loaded. The only geometry editing functions are for rotating the vehicle about one of the axes or about the normals for selected geometry panels. See Section 1.2.4 for documentation of the geometry viewer.
- ❑ **Save Geometry** – saves the geometry in a specified file. This is required if the geometry has been edited (i.e., flipped about one of its axes or normals).



**Figure 2 Main Window - Geometry Analysis**

Figure 3 shows the tabbed pane for “Aero Analysis”. It consists of buttons that provide the following functions:

- ❑ **Streamlines** – brings up a workflow window for generating streamlines using Quadstream. This is only required when doing “Level 2 Viscous” high-speed aero analysis. See Section 3 for documentation of the Quadstream workflow window.
- ❑ **HighSpeed Aero** – brings up a workflow window for doing high-speed aero analysis using S/HABP. See Section 4 for documentation of the S/HABP workflow window.

- **LowSpeed Aero** – brings up a workflow window containing a plug-in for doing low-speed aero analysis. See Section 5 for documentation of the low-speed aero plug-in.

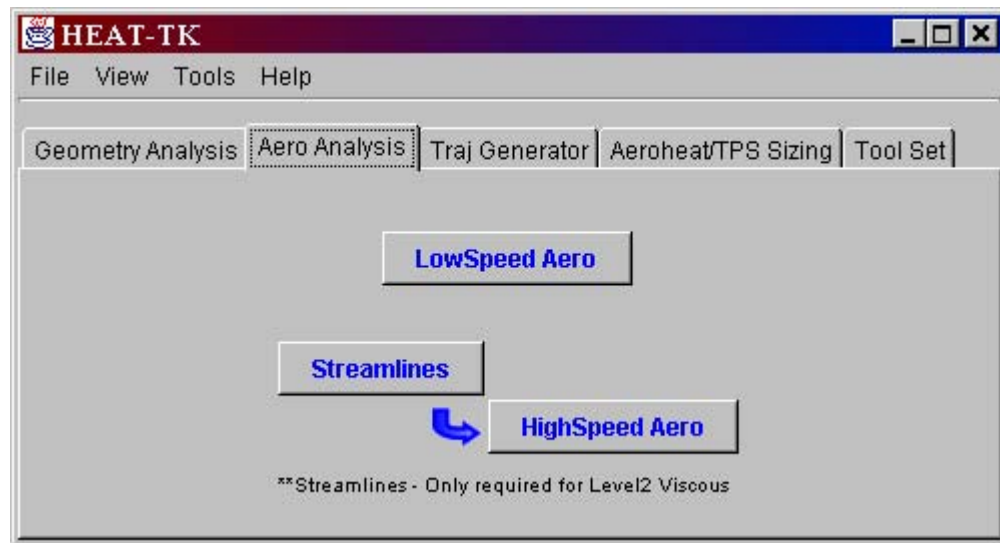


Figure 3 Main Window - Aero Analysis

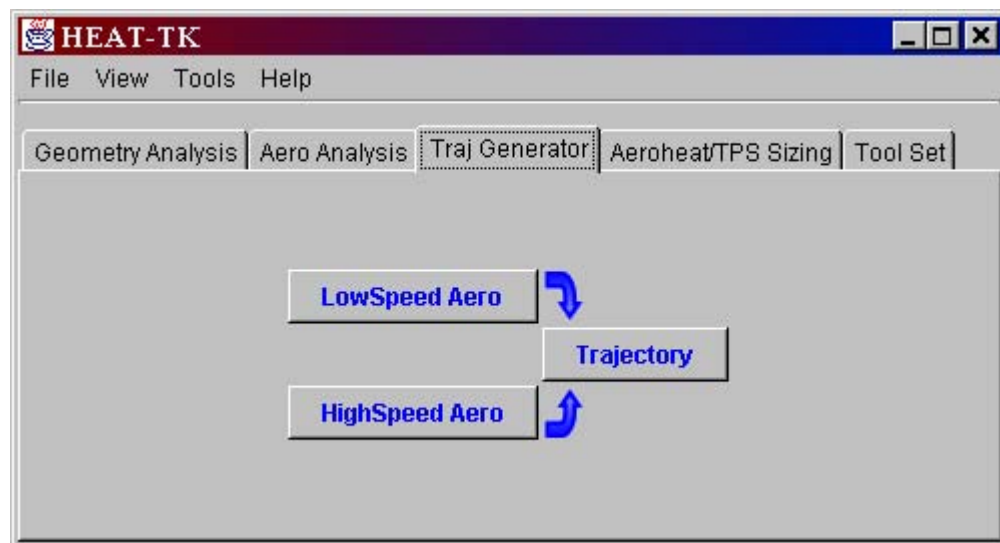
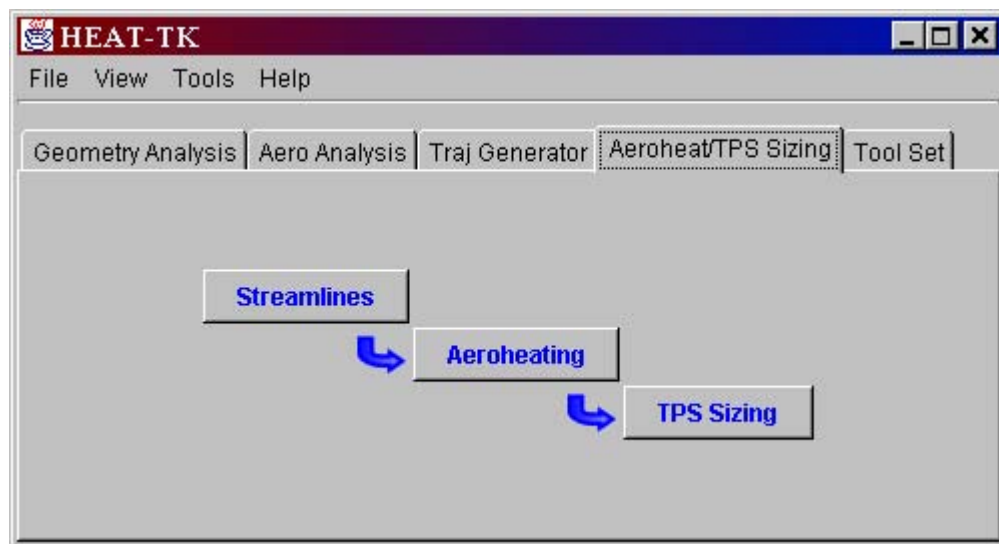


Figure 4 Main Window - Trajectory Generator

Figure 4 shows the tabbed pane for “Trajectory Generator”. It consists of buttons that provide the following functions:

- **LowSpeed Aero** – brings up a workflow window containing a plug-in for doing low-speed aero analysis. See Section 5 for documentation of the low-speed aero plug-in.

- ❑ **HighSpeed Aero** – brings up a workflow window for doing high-speed aero analysis using S/HABP. See Section 4 for documentation of the S/HABP workflow window.
- ❑ **Trajectory** – brings up a workflow window containing a plug-in for doing aero analysis along a trajectory. See Section 6 for documentation of the trajectory plug-in. Note that at least one of the low-speed or high-speed aero analysis functions must be performed prior to executing the trajectory plug-in.



**Figure 5 Main Window - Aeroheat/TPS Sizing**

Figure 5 shows the tabbed pane for “Aeroheat/TPS Sizing”. It consists of buttons that provide the following functions:

- ❑ **Streamlines** – brings up a workflow window for generating streamlines using Quadstream. See Section 3 for documentation of the Quadstream workflow window.
- ❑ **Aeroheating** – brings up a workflow window for doing trajectory aeroheating analysis using S/HABP. See Section 7 for documentation of the trajectory aeroheating workflow window.
- ❑ **TPS Sizing** – brings up a workflow window for performing TPS sizing analysis. See Section 8 for documentation of the TPS sizing workflow window.

Figure 6 shows the tabbed pane for “Tool Set”. It includes buttons for all of the analysis tools in HEAT-TK. The only buttons that do not appear on at least one of the other tabbed panes are:

- ❑ **3D Viewer** – displays a specified VRML file in the 3D viewer. See Section 1.2.4 for documentation of the 3D viewer.
- ❑ **2D Plotter** – brings up the 2D plotter Ptplot. See Section 1.2.3 for documentation of Ptplot.

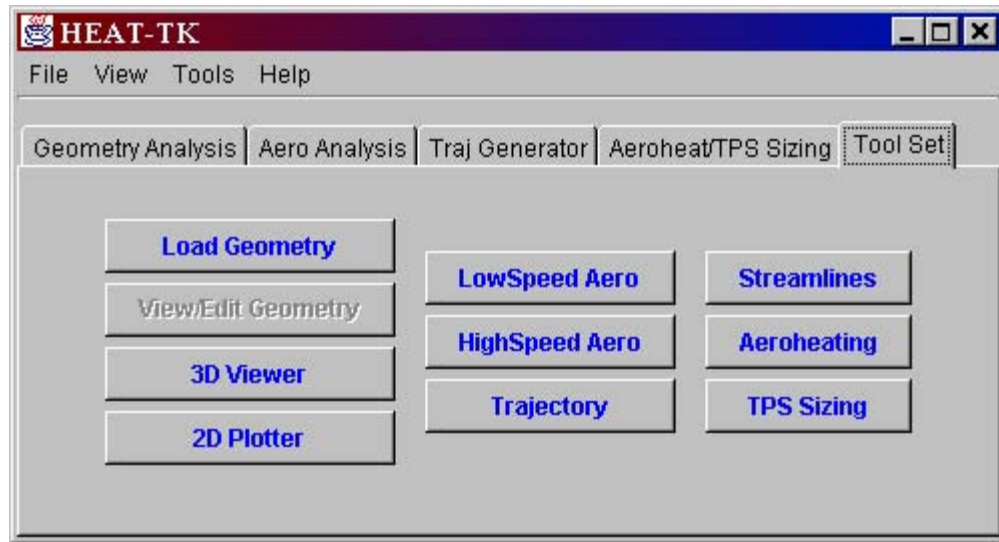


Figure 6 Main Window - Tool Set

### 1.2.2 Workflow Window

A workflow window encapsulates a sequence of steps related to a specific aerothermal analysis process. The sequence of steps typically includes loading file(s), setting input values, running an analysis, and viewing results. As mentioned in the previous section, there are separate workflow windows for each of the following processes: Streamlines, HighSpeed Aero, LowSpeed Aero, Trajectory, Aeroheating, and TPS Sizing. Each of these workflow windows has a similar layout and approach, as discussed in the remainder of this section.



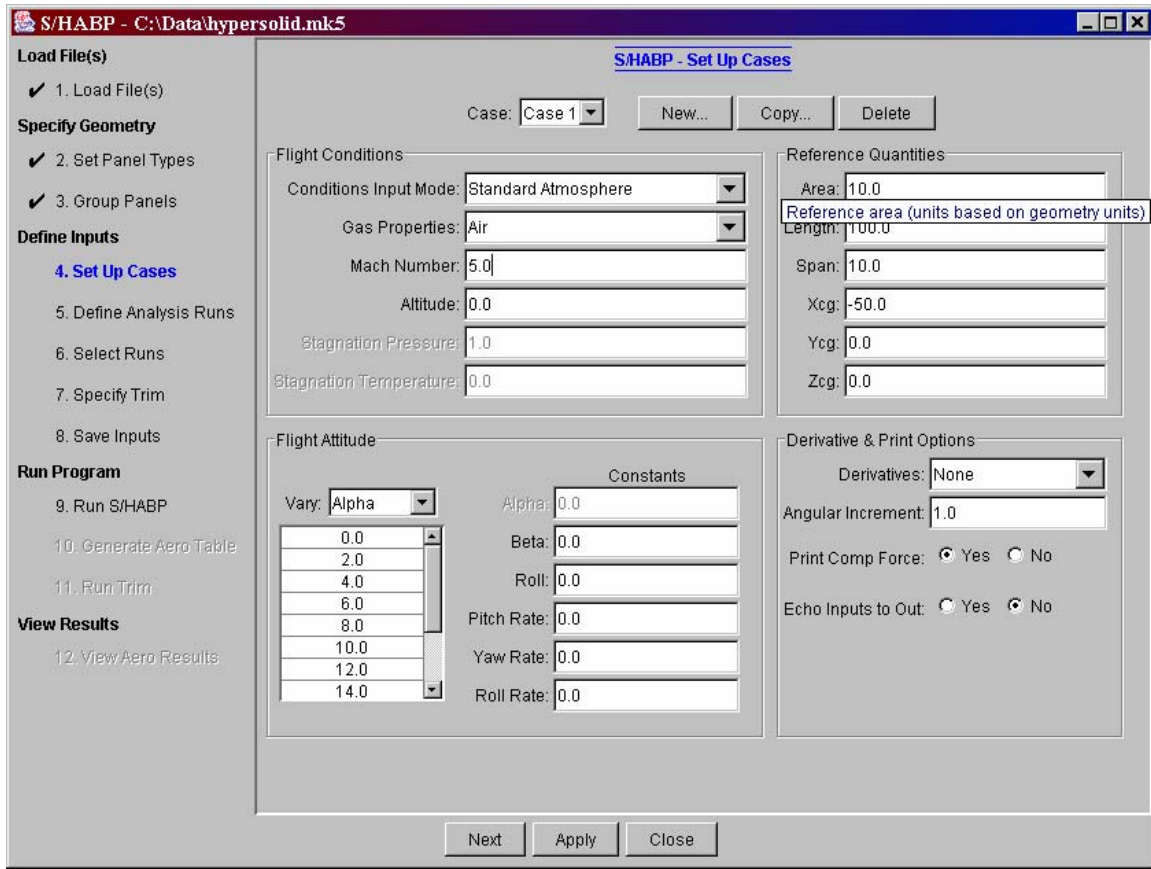


Figure 7 Sample Workflow Window

Figure 7 is taken from one of the steps in a HighSpeed Aero workflow window and will serve as an example. Every workflow window is broken up into 3 panels:

1. **Process steps** – a panel on the left side of a workflow window contains the sequence of steps applicable to the process. These steps are numbered and may be grouped into categories. Individual steps can be clicked on and information related to that step appears in the main “work panel.” The step currently being worked on is highlighted in blue, other “available” steps are displayed in black, and “unavailable” steps are grayed out. When a step has enough information to be considered “complete”, it has a checkmark placed next to it, and as a result, additional steps may become “available”. Changing information in an earlier step may render later step(s) “incomplete”, thus removing checkmark(s). Note that the workflow process “favors completing steps”. For example, clicking on a step later in the process than the current step will trigger the workflow process to save information for each step prior to the newly requested step so that they will be considered “complete”. This is done whenever possible. In the example workflow window, the first 3 steps have been completed, and step 4 (“Set Up Cases”) is currently being worked on. Information related to any of the first 9 steps can be viewed, although steps 5-9 are incomplete. Also, steps 10, 11, and 12 are not yet available.
2. **Work panel** – the recessed panel to the right of the process steps (taking up the majority of the workflow window) is used to display and edit information related to the current step. This

information may be related to loading files, setting input parameters, saving files, running programs, or viewing results, depending on the step selected.

*Tip:* Many of the data items that can be set in the “work panel” have one-line help descriptions available. To access a one-liner, hold the mouse pointer over a data item name. In the example workflow window, the “work panel” displays information related to step 4 (“Set Up Cases”). In this panel, input settings related to “cases” can be set, including flight conditions, flight attitude, reference quantities, and derivative and print options. The one-line help description for “Area” was brought up by holding the mouse over the word “Area” – the one-liner appears in the white box in the upper right “on top of” the work panel window.

3. **Button panel** – the buttons at the bottom of the workflow window provide some additional functionality. The “Next” button is used to go to the next step in the workflow sequence. In clicking this button, information for the current step is automatically “applied”. (Note that the next step can also be directly selected from the list of steps in the “Process steps” panel.) The “Apply” button is used to save information from the current step. The “Close” button closes the workflow window. If any information needs to be saved to a file before closing, an option to do this is first presented prior to closing the window.

There are separate workflow windows for each of the aerothermal analysis processes, and these are documented in the following sections:

- ❑ **Streamlines** – for generating streamlines using Quadstream; see Section 3.
- ❑ **HighSpeed Aero** – for doing high-speed aero analysis using S/HABP; see Section 4.
- ❑ **LowSpeed Aero** – contains a plug-in for doing low-speed aero analysis; see Section 5.
- ❑ **Trajectory** – contains a plug-in for doing aero analysis along a trajectory; see Section 6.
- ❑ **Aeroheating** – for doing trajectory aeroheating analysis using S/HABP; see Section 7.
- ❑ **TPS Sizing** – for performing TPS sizing analysis; see Section 8.

### 1.2.3 Two-D Plot Window

HEAT-TK provides 2D data plots using Ptplot, an open-source Java plotting package from the University of California at Berkeley. Ptplot has the following basic 2D plotting capabilities:

- ❑ multiple sets of data
- ❑ marking data using various combinations of lines, color, and symbols
- ❑ zooming in on regions of a plot
- ❑ changing plot titles and axis labels
- ❑ saving plots to XML
- ❑ printing plots (or printing to a postscript file)

The plotter is normally brought up when viewing 2D results from an aerothermal analysis. However, it can also be accessed from the main window via the “2D Plotter” button. A sample 2D plot window is shown in Figure 8. The “plot controls” window is shown in Figure 9 and can be accessed via the “Edit => Format” menu option of Ptplot. Further documentation is available on the Ptplot web site, <http://ptolemy.eecs.berkeley.edu/java/ptplot/> (after selecting a version, e.g., Ptplot 5.2, click on “Plot Chapter” for the main documentation related to using the plotting package). This documentation can also be accessed from the HEAT-TK main window help menu.

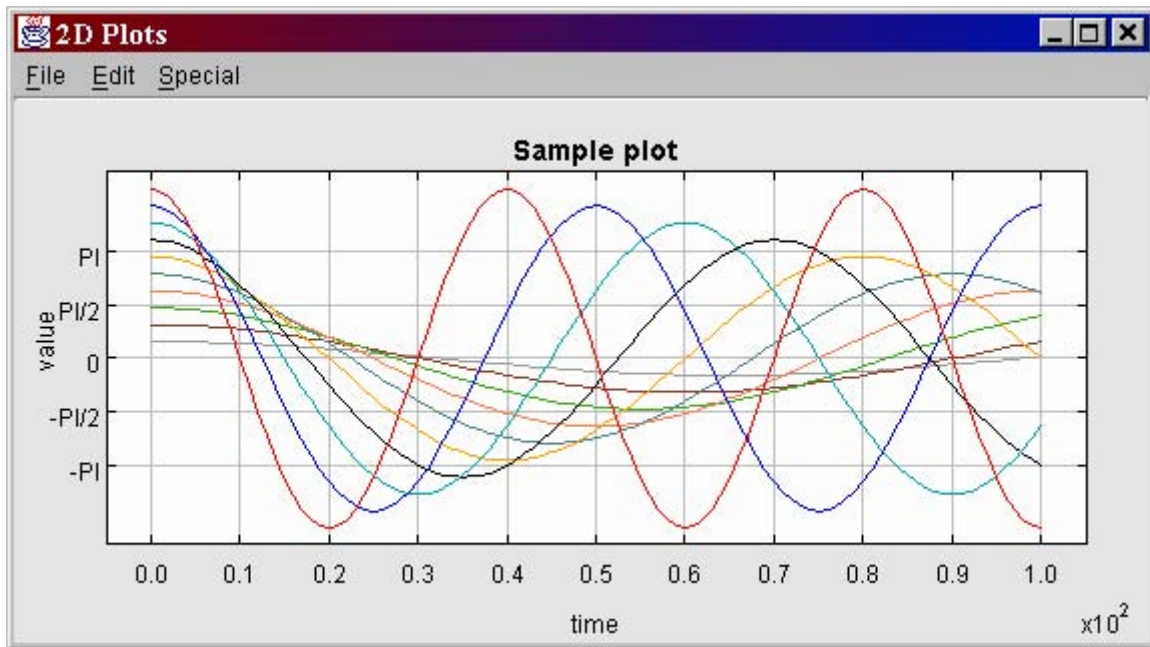


Figure 8 Sample 2D Plot Window

Figure 9 Plot Controls Window

### 1.2.4 Three-D VRML Viewer

HEAT-TK uses a web browser plug-in to provide 3D viewing capability. A 3D surface is represented in VRML (Virtual Reality Modeling Language), and the VRML file is loaded in a web browser. If the browser plug-in for displaying VRML files is installed, then the 3D surface

can be examined via rotating, translating, and zooming. Cosmo player is the browser plug-in that must be installed; see the “Installation Instructions” for more information.

The 3D viewer is accessed from HEAT-TK in one of the following ways:

- ❑ From the main window via the “3D Viewer” button. In this case, a VRML file must be specified.
- ❑ When viewing a 3D surface in the course of doing aerothermal analysis. In this case, one or more parameters must first be selected to specify how to display the 3D surface. The particular parameters to specify depend on whether the view is of geometry panels/components, or a distribution mapped onto the surface (e.g., Cp), or streamlines, ... Figure 10 provides an example of parameters that can be specified in viewing geometry panels/components. After the parameters are specified, click on the “View” button to automatically bring up a web browser with the 3D surface displayed. The most common parameters are described next.

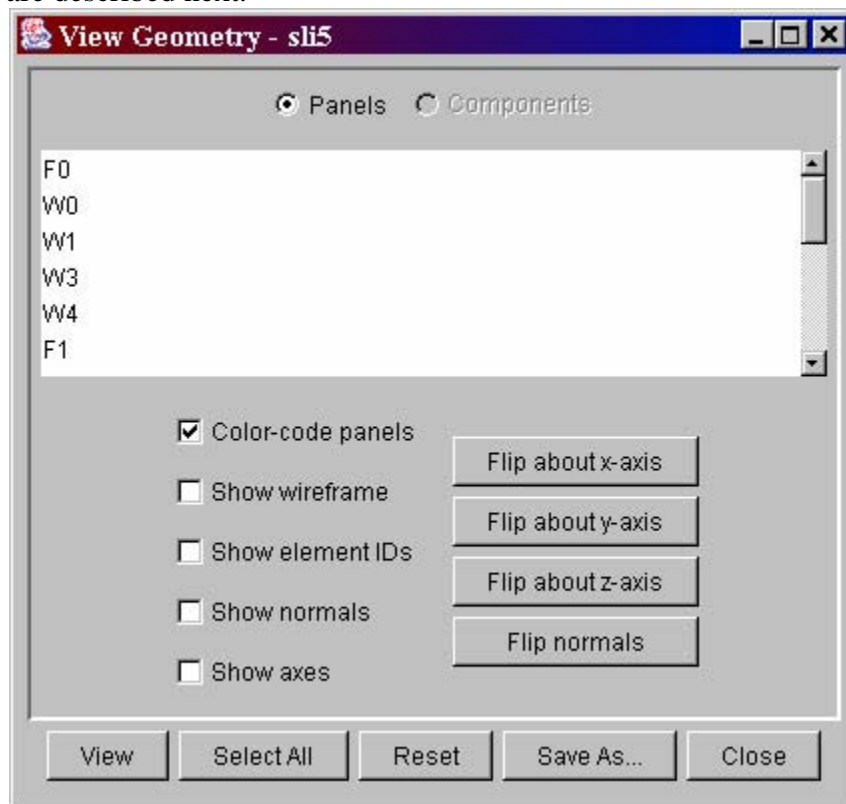


Figure 10 Sample 3D Controls Window

### 3D Control Parameters

The control parameters that specify how to display a 3D surface include:

- ❑ **Color-code panels** – specifies whether to color-code geometry panels/components. If checked, different panels will be different colors. If not checked, all panels will be gray.
- ❑ **Show wireframe** – specifies whether to display a wireframe outlining the geometry elements that is offset from the surface.

- ❑ **Show element Ids** – specifies whether to include a ‘mouseover geometry element’ feature. If checked, information about a given geometry element will be displayed in the lower left corner of the browser when the mouse is pointing to that element. This information includes the panel name, element #, centroid, and, if available, a mapped scalar value (e.g., Cp). If not checked, only a panel/component name is displayed.  
Note: there is sometimes a problem showing element info when ‘show wireframe’ is also checked. Unchecking ‘show wireframe’ corrects this problem.
- ❑ **Show normals** – specifies whether to include normal spikes at the center of each geometry element.
- ❑ **Show axes** – specifies whether to include (x,y,z) axes in the view. The axes rotate with the surface.

In addition to these control parameters, there is an option to view selected geometry panels and components (if components are defined). If the “Panels” radio button at the top of the window is selected, then a list of geometry panels appears below the radio buttons. Geometry panels for viewing can be selected via:

- ❑ Clicking on an individual geometry panel,
- ❑ Holding down the <ctrl> key and clicking on a panel to include it to the selected list,
- ❑ Holding down the <shift> key and clicking on a panel to include a sequence of panels in the selected list, or
- ❑ Clicking on the “Select All” button to select all of the geometry panels.

After selecting one or more geometry panels, click the “View” button to display the selected panels. A similar process can be performed for geometry components after clicking on the “Components” radio button.


There are also buttons on the “Controls Window” to perform various functions related to the geometry view. The buttons that are available depend on whether the view is of geometry panels/components, or a distribution mapped onto the surface (e.g., Cp), or streamlines, ... The buttons are a subset of the following:

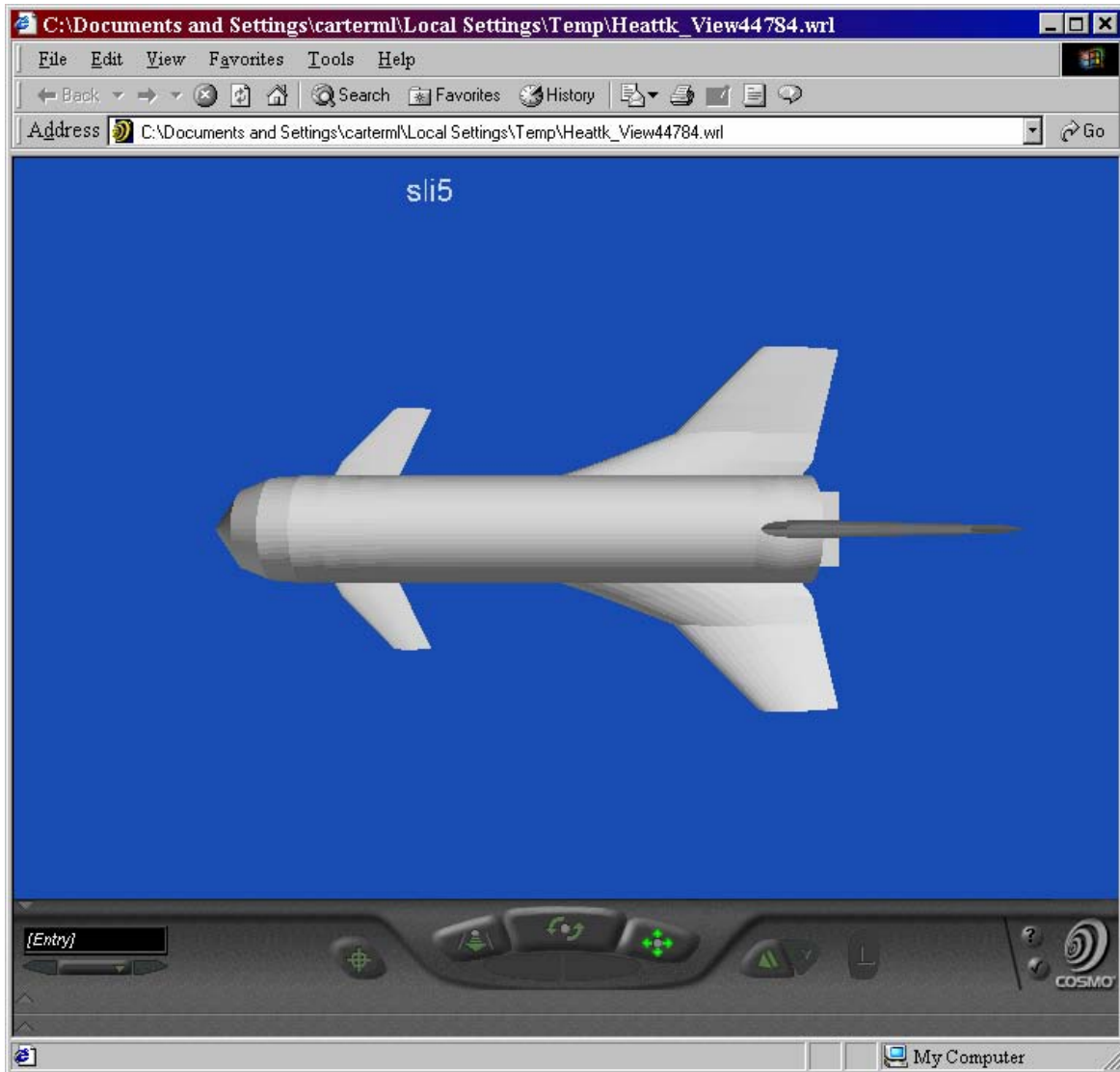
- ❑ **View** – displays the geometry, according to the specified control parameters, in a web browser (see below for further description)
- ❑ **Select All** – selects all the geometry panels/components in the list for viewing
- ❑ **Reset** – if a geometry has been changed via one of the “Flip” buttons, the Reset button resets the geometry to its original unaltered state
- ❑ **Save As** – saves a VRML file containing the current geometry view. This file can be viewed later in a web browser. Note that this does **not** save a geometry that has been changed via one of the “Flip” buttons – this is done via the “Save Geometry” button on the main HEAT-TK window.
- ❑ **Close** – closes the controls window
- ❑ **Flip about x-axis** – flips the selected panels/components for the current geometry about the x-axis
- ❑ **Flip about y-axis** – flips the selected panels/components for the current geometry about the y-axis
- ❑ **Flip about z-axis** – flips the selected panels/components for the current geometry about the z-axis

- ❑ **Flip about normals** – flips the selected panels/components for the current geometry about their normals


### 3D Viewer Window

A sample 3D viewer window is shown in Figure 11. The center part of the window contains a view of the surface. The “control panel” at the bottom of the browser window provides buttons to rotate, translate, and zoom in on the 3D surface. To use one of these features, click on one of

the buttons in the control panel. For example, click on the “rotate” button:  Then, move the mouse in the center part of the window, and, holding down the right mouse button, drag it until the desired surface rotation is displayed. This “click and drag” motion may need to be done more than once to achieve the desired view.



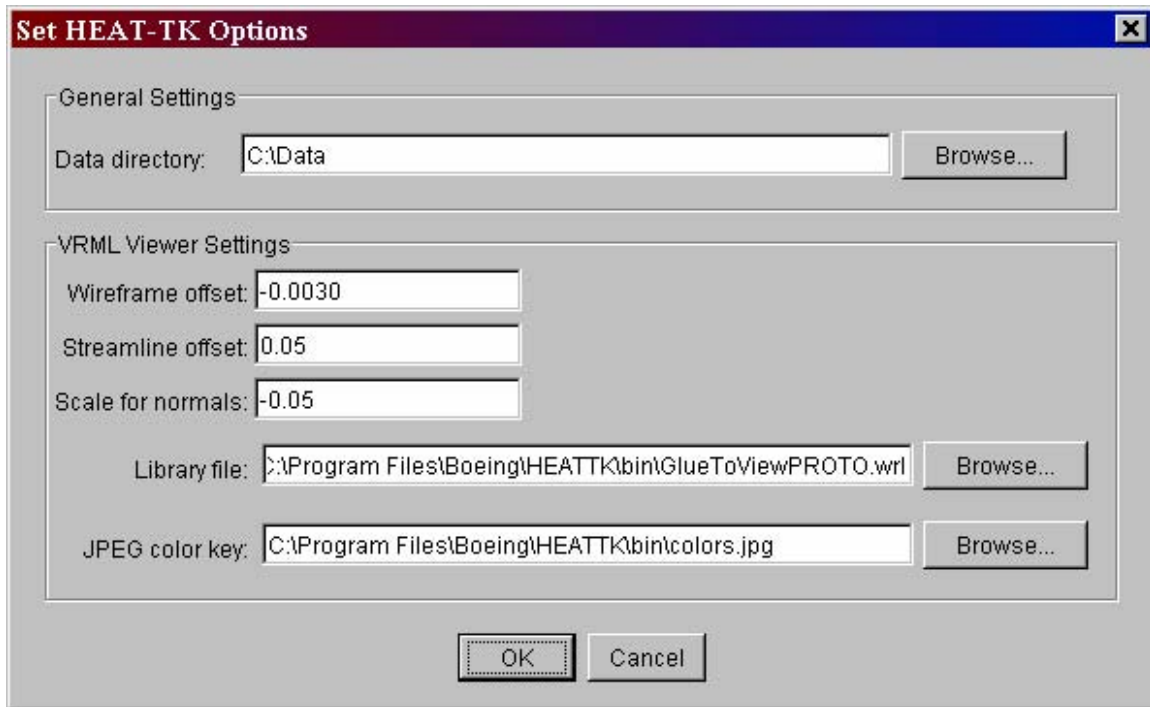
**Figure 11 Sample 3D Viewer Window**

Further documentation on using the 3D viewer is available online. The “help icon”  appears on the right portion of the “control panel” when viewing a 3D surface in the browser. Clicking on this icon brings up a web page with online help. This documentation can also be accessed from the HEAT-TK main window help menu. In addition, the Cosmo Player plug-in web site, <http://www.ca.com/cosmo/>, provides some documentation.

### **1.2.5 HEAT-TK Parameter Settings**

Several parameters, mostly related to the 3D VRML viewer, can be set by selecting the “Tools => Options” menu item. The resulting window is shown in Figure 12.





**Figure 12 HEAT-TK Parameter Settings**

The parameters that can be set from this window apply to the current session of running HEAT-TK; there is no mechanism to save parameter settings to a file. Here is a brief description of the parameters (see Section 1.2.4 for additional description of the 3D VRML Viewer):

- ❑ **Data directory** – a directory where HEAT-TK looks for data files, specifically when a “Browse” button is clicked.
- ❑ **Wireframe offset** – when the “Show wireframe” option is selected in viewing a 3D surface, a wireframe is displayed, offset from the surface. The ‘wireframe offset’ parameter controls how far the wireframe is offset from the surface. A negative value should be used, and values with a larger absolute value result in a wireframe further from the surface. The default value is  $-0.003$ ; a recommended range is  $[-0.05, -0.0005]$ .
- ❑ **Streamline offset** – when streamlines are being viewed in the 3D viewer, they are offset from the surface. The ‘streamline offset’ parameter controls how far the streamlines are offset from the surface. A positive value should be used, with larger values resulting in streamlines further from the surface. The default value is  $0.05$ ; a recommended range is  $[0.01, 1.0]$ .
- ❑ **Scale for normals** – when the “Show normals” option is selected in viewing a 3D surface, ‘spikes’ are displayed normal to the surface at each geometry element. The ‘scale for normals’ parameter controls the length of the spikes. A negative value should be used, and values with a larger absolute value result in larger spikes. The default value is  $-0.05$ ; a recommended range is  $[-0.1, -0.01]$ .
- ❑ **Library file** – when a VRML file is generated on the fly for viewing a 3D surface, it relies on a special-purpose VRML library file. The default location for this file is



*<install\_directory>\HEATTK\bin\GlueToViewPROTO.wrl* . This value should only need to be changed if a VRML file generated by HEAT-TK is to be viewed over the web or sent to another analyst. In this case, storing a copy of the library file in a central location and setting the value for this 'library file' parameter to that location (e.g., an http or file server location).

- **JPEG color key** – this parameter is similar to the 'library file', only it is a JPEG file containing a color key. The default location for this file is *<install\_directory>\HEATTK\bin\colors.jpg*.

## 2 Geometry Modeling

The geometry model used by HEAT-TK is the same as the one in VECC. It consists of the x-, y-, and z-coordinates of points that make up the vehicle's outer mold line (OML). The vehicle OML is subdivided into components, panels, sections, and elements. A geometry model is stored in the "geo" format. Additional description of the geometry model is provided in Section 2 of the VECC Software User's Manual. This section documents the usage of IGES files to import OML into HEAT-TK, and the process of setting attributes of geometry panels and their grouping.

### 2.1 IGES Files

HEAT-TK can accept vehicle OML as an Initial Graphics Exchange Specification (IGES) file. IGES version 5.3 (ANS US PRO/IPO-100-1996) specifies a text file format commonly used for passing geometry data between applications. When loading a geometry file, if it has an ".igs" or ".iges" file extension, then HEAT-TK assumes it to be an IGES file and automatically converts it to a "geo" geometry model in a 4-step process. If the conversion succeeds, then the geometry can be used for performing further analysis.

HEAT-TK uses a restricted subset of IGES format to import vehicle OML and convert it to a "geo" geometry model. In particular, HEAT-TK only uses four of the many IGES entity types, namely types 143, 141, 128 and, 126. A brief description of the general IGES format and the format for these four entity types are given in Section 2.1.2.

Section 2.1.1 contains a description of the additional restrictions imposed on IGES files by HEAT-TK.

#### 2.1.1 Additional IGES Restrictions Imposed by HEAT-TK

The capability of HEAT-TK IGES processor is very limited. However, for certain geometry shapes, such as that of a flight vehicle, it can be used successfully to import IGES geometry into HEAT-TK. Its limitations include the following:

- All IGES entities except 126, 128, 141, and 143 are ignored.
- The names of IGES entities are used to determine their discretization method.
- The number of grid points used to discretize each entity is fixed.
- Trimming curves for entities named to be wing or body are ignored, i.e., the untrimmed surface is discretized.
- Trimming curves for arbitrary surface patches are assumed to be parallel to the coordinate axes in the parametric space.

In order to properly discretize surfaces, HEAT-TK examines the names of the IGES entities. If the name of an entity contains one (or more) of the substrings "wing," "WNG," "bflap," "canard," or "tail," then the entity is processed as a wing. This means the entity is discretized into up to six airfoils with 18 points per airfoil. If the name of an entity contains one (or more) of the substrings "body," "BDY," or "fair," then the entity is processed as a body. A body is sliced into 20 cross sections at constant axial stations with 26 points per section. If the entity name does not conform to these conventions, it is processed as an arbitrary surface patch. In this case, it is discretized into a 20 by 20 grid.

Although HEAT-TK reads type 143 entities, the information contained in the entity is mostly ignored. If the entity represents a wing or a body, then the information is completely ignored. If the entity represents an arbitrary surface patch, then the trimming curves must be parallel to the coordinate axes in parametric space, i.e. the trimmed region must represent a sub-rectangle of the rectangle over which the entire surface is defined, and the edges of the sub-rectangle must be parallel to the edges of the original surface in parametric space.

### **2.1.2 Description of IGES Entities Used by HEAT-TK**

IGES defines several formats, but the following description outlines the one, which is so commonly used that it is the only one that could be meant by “the IGES format”. IGES files consist of 80-character lines of ASCII graphic characters. Each line terminates with an 8-character sequence field consisting of a capital letter identifying the “section” to which the line belongs and a right-justified line number in a 7-character field. The line numbers are consecutive from one in each section. The sections appear in a fixed order as listed below. The five “sections” are:

- S Start section. Human readable documentation in columns 1 to 72.
- G Global section. Up to twenty-six free-form, comma-delimited, semicolon-terminated, defined pieces of information describing the file as a whole in columns 1 to 72 of each line.
- D Directory section. Two lines for each entity. Ten 8-character fields in each line.
- P Parameter section. The rest of the data for each entity in a free-form, comma-delimited, semicolon-terminated, format within columns 1 to 64 plus a reference back to the corresponding Directory section line in columns 65 to 72.
- T Terminate section. One line listing the total numbers of lines in each of the preceding four sections.

It is possible to use the first two global parameters to change the punctuation characters used for comma and semicolon, but doing this is not advised.

IGES defines numerous entity types and identifies each one with a positive integer called the IGES entity type number. Some entity types are divided into subtypes via a “form” number. The combination of type number and form number determine the interpretation of the sequence of fields that appear for that entity in the Parameter section. When an IGES entity needs to make reference to another IGES entity, the reference is made by listing the line number of the first line of the Directory section for the referenced entity.

The fields in the two lines of each entity in the Directory section contain the following information:

First (odd-numbered) line:

1. Entity type number
2. Line number of first line of corresponding data in Parameter section
3. Structure \*
4. Line Font Pattern \*

5. Level \*
6. View \*
7. Reference to associated transformation matrix entity, if any
8. Label Display Assoc \*
9. Status Number (Four two-digit subfields with fixed meanings) \*
10. The required sequence field, including the number by which this entity is referenced from elsewhere

Second (even-numbered) line:

1. Entity type number (repeat from same field in previous line)
2. Line Weight Number \*
3. Color Number \*
4. Number of associated lines in Parameter section
5. Form Number
6. Unused (and reserved)
7. Unused (and reserved)
8. Entity Label (up to 8-character identifier for entity)
9. Entity Subscript Number (up to 8-digit additional identification for entity)
10. The required sequence field

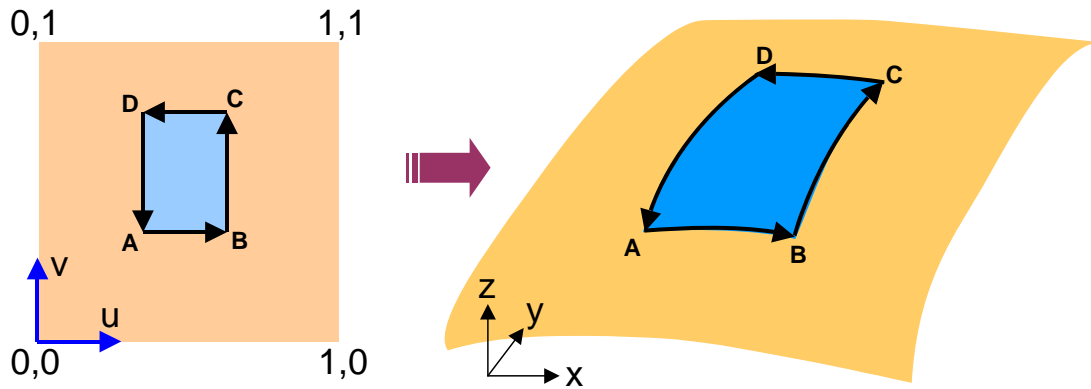
\* See IGES standard for more information

The four IGES entity types used by HEAT-TK are 143, 141, 128 and 126. The contents of the fields in the Parameter section of each of these entity types follow.

### **Type 143**

IGES type 143 is formally named the “Bounded Surface Entity” and also commonly known as the “Trimmed Surface” entity. An alternative with roughly equivalent content is the historically older IGES type 144 formally named “Trimmed (Parametric) Surface Entity”. Trimmed surfaces are made up of two components. The first is a mapping from a given rectangular region in the 2D plane to a surface in 3D via three coordinate functions, which map each of the points in the 2D rectangle to the three coordinates of a point in 3D. The second is a collection of curves, defined either in the parametric domain or on the surface itself, which define the portion of the surface, which is active, as shown in blue in Figure 15. HEAT-TK discretizes the region bounded by the curves (AB, BC, CD, and DA). Only four bounding curves are allowed and they should all be oriented counterclockwise. HEAT-TK imposes the further restriction that the four curves be parallel to the  $u$  and  $v$  axes in parameter space (the left picture above). The fields in the Parameter section for an entity of IGES type 143 are:

1. Flag indicating whether the remaining data includes parametric trimming curves or not (0 = no, 1 = yes).
2. Pointer to the underlying surface, which must be a type 128 entity for HEAT-TK.
3. The number of trimming boundaries.
4. The first of the pointers to IGES type 141 entities serving as trimming boundaries for this entity (additional ones are stored in subsequent fields).



**Figure 13 Type 143 Entity in Physical and Parametric Space.**

### **Type 141**

IGES type 141 is formally named the “Boundary Entity”. The corresponding boundary for IGES type 144 is IGES type 142. The fields in the Parameter section for IGES type 141 are:

1. Flag indicating whether the remaining data includes parametric trimming curves corresponding to each space trimming curve or not (0 = no, 1 = yes).
2. Code indicating preferred representation for trimming curves in sending system (0 = unspecified, 1 = model space, 2 = parameter space, 3 = no preference).
3. Pointer to underlying surface.
4. Number of space curves included in this boundary.
5. The first of the space curves (additional ones are stored in subsequent fields):
  - a. Pointer to space curve.
  - b. Flag indicating whether space curve is used as is, or reversed (0 = as is, 1 = reversed)
  - c. Number of associated parametric curves, if any
  - d. The first of the pointers to parametric curves, in order (additional parametric curves associated with the given space curve are stored in subsequent fields).

The z coordinates of parametric curves are ignored. The fact that they are actually 2D parametric curves rather than 3D space curves should also be indicated by a “05” in the third subfield of the Status Number in the corresponding Directory section line. Because the surface may have seams or poles, parametric curves corresponding to a space curve boundary need not be connected or collectively form a closed curve in parameter space even when the space curves collectively form a closed curve.

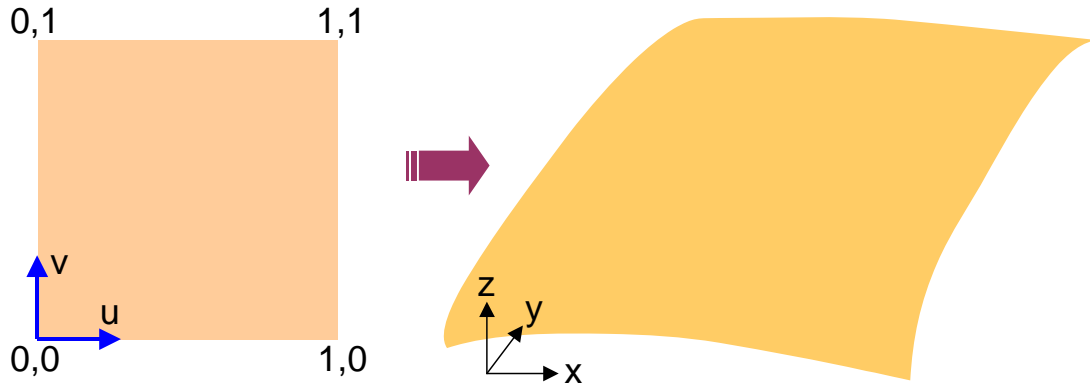


Figure 14 Type 128 Entity in Physical and Parametric Space

### Type 128

IGES type 128 is formally named the “Rational B-Spline Surface Entity,” but this is something of a misnomer, as the actual entity can be used to represent all tensor product rational spline surfaces. Tensor product spline surfaces are defined as mappings from a given rectangular region in the 2D plane to surfaces in 3D via three coordinate functions which map each of the points in the 2D rectangle to the three coordinates of a point in 3D, as shown in Figure 16. HEAT-TK discretizes the entire surface. The form number provides information about the geometric properties of the surface and does not affect the format of the data. The fields in the Parameter section are:

1. One less than the number of basis functions in the first parameter.
2. One less than the number of basis functions in the second parameter. The total number of control points is the product of the numbers of basis functions in each parameter.
3. The degree of the basis functions in the first parameter.
4. The degree of the basis functions in the second parameter.
5. Flag for being closed in the first parameter direction (0 = no, 1 = yes).
6. Flag for being closed in the second parameter direction (0 = no, 1 = yes).
7. Flag for being rational or polynomial (0 = rational, 1 = polynomial).
8. Flag for being periodic in the first parameter (0 = no, 1 = yes).
9. Flag for being periodic in the second parameter (0 = no, 1 = yes).
10. Knots of the first independent variable. The number of knots is given by the number of basis functions in the first variable plus the degree of the basis functions in that variable plus one.
11. Knots of the second independent variable. The number of knots is given by the number of basis functions in the second variable plus the degree of the basis functions in that variable plus one.
12. Weights for control points. The number of control points is equal to the product of the number of basis functions in each variable.
13. Coordinates of control points in 3D. The control points are listed consecutively, and their number is equal to the product of the number of basis functions in each variable.
14. Starting value of domain in first parameter.

15. Ending value of domain in first parameter.
16. Starting value of domain in second parameter.
17. Ending value of domain in second parameter.

### **Type 126**

IGES type 126 is formally named the “Rational B-Spline Curve Entity”. The form number provides information about the geometric properties of the curve and does not affect the format of the data. The fields in the Parameter section are:

1. One less than the number of basis functions in the first parameter. The number of control points is the same as the number of basis functions.
2. The degree of the basis functions.
3. Flag for being planar (in space) (0 = no, 1 = yes).
4. Flag for being closed (0 = no, 1 = yes).
5. Flag for being rational or polynomial (0 = rational, 1 = polynomial).
6. Flag for being periodic (0 = no, 1 = yes).
7. Knots of the spline. The number of knots is equal to the number of control points plus the polynomial degree plus one.
8. Weights of the spline. The number of weights is equal to the number of control points.
9. Coordinates of the control points. Each control point has three coordinates. The control points are listed in order. The number of control points is one more than the number given by parameter 1.
10. Starting value of parameter domain.
11. Ending value of parameter domain.
12. Coordinates of unit normal when curve is planar.

For each entity type, IGES permits two groups of parameters in addition to the listed parameters. Each group has the form of a count followed by that many entries. These additional groups of parameters are not permitted by HEAT-TK.

## **2.2 Set Panel Attributes**

Several HEAT-TK workflow windows include a step to “Set Panel Types”, which allows a user to specify units for the geometry and attributes for each geometry panel. Figure 15 shows a sample window featuring the “Set Panel Types” step. This functionality was provided and documented in VECC – see Sections 2.5 and 2.6 of the VECC manual for additional information.

Note: For aeroheating and TPS sizing analysis, users should consider setting up specific panels based on the geometry of the vehicle, heating methods applied and TPS material selection. For example, separate panels may be set up for the wing leading edge (LE), lower and upper wing surfaces. These locations require modeling with different heating methods. Also different TPS materials may be used on LE panels as compared to the flat acreage locations.

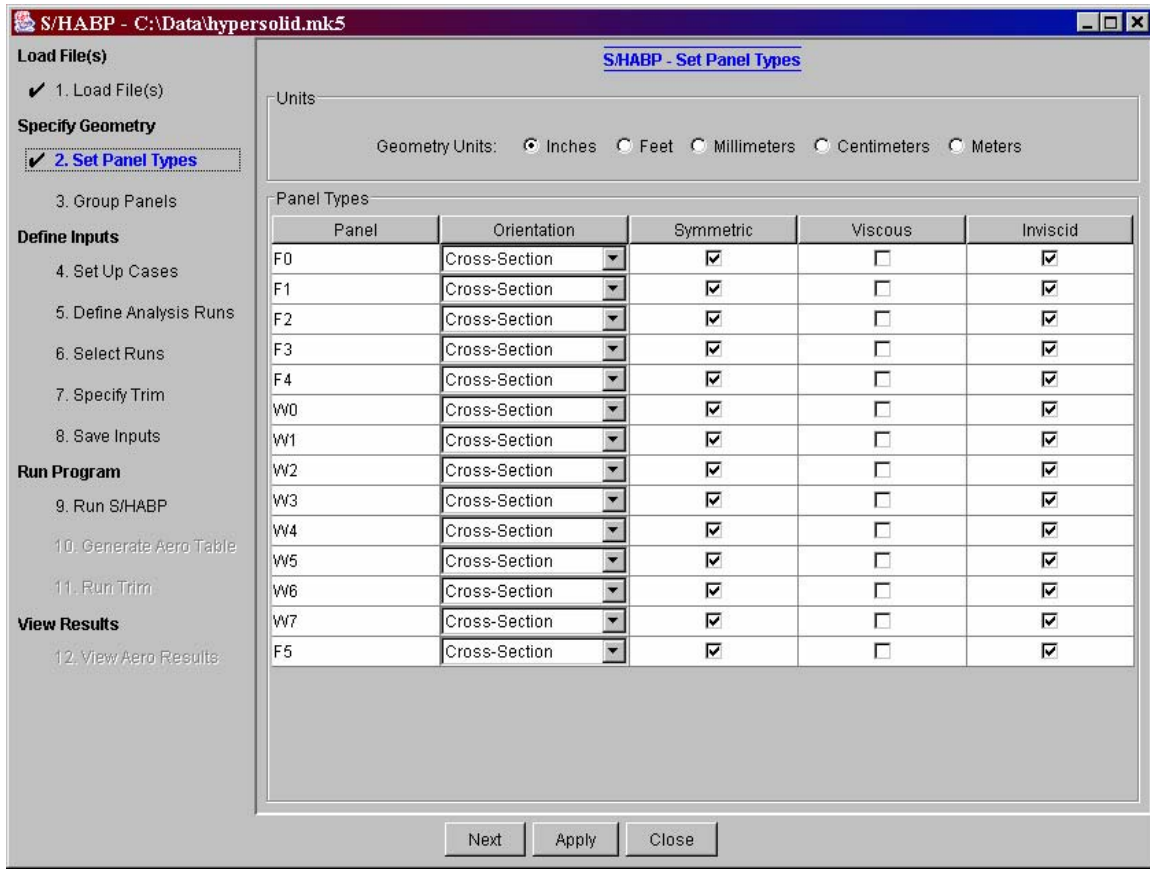


Figure 15 Sample "Set Panel Types" Window

## 2.3 Grouping panels

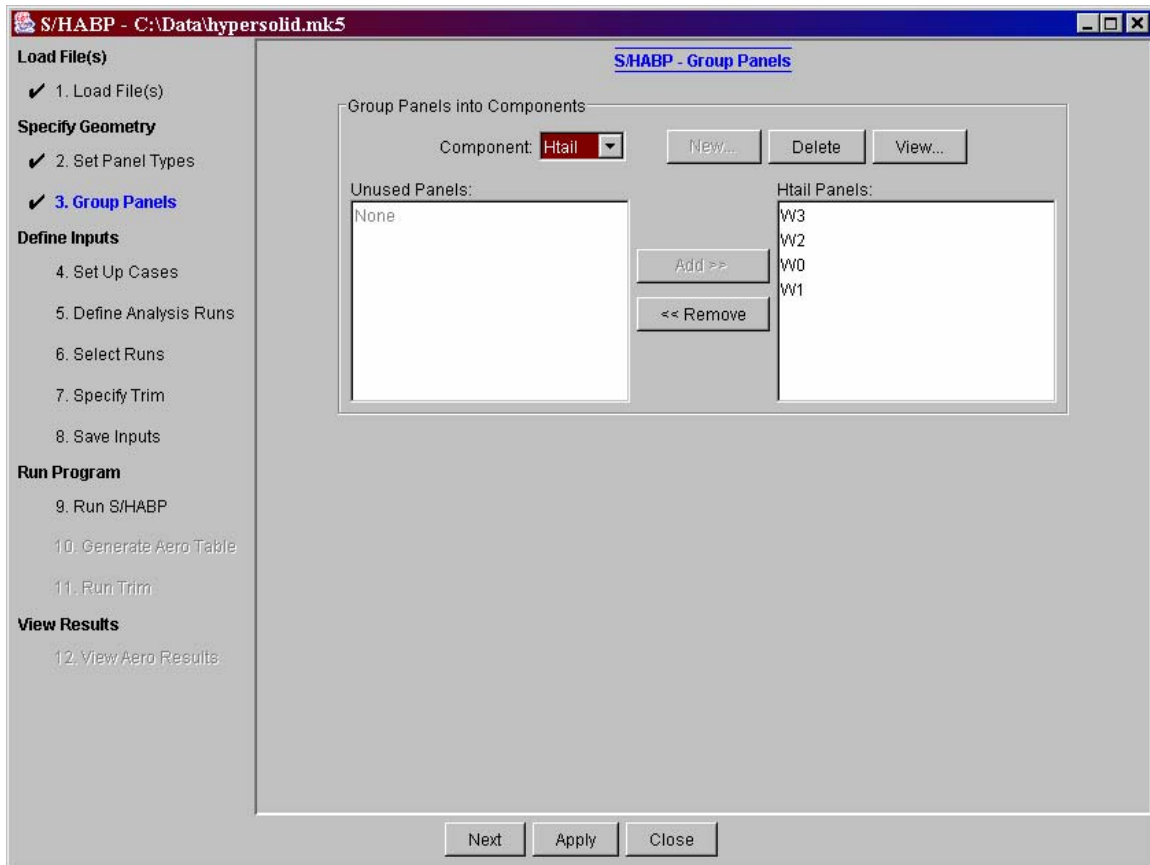
Several HEAT-TK workflow windows include a step to “Group Panels”, which allows a user to group geometry panels to form components. Geometry components are major parts of a vehicle that are to be analyzed as a unit (i.e., a wing, tail, etc.). Figure 16 shows a sample window featuring the “Group Panels” step.

A list of geometry components already defined for this vehicle is given in the pull down list. A new geometry component is created by clicking on the “New” button and specifying a name. If all geometry panels are already part of a component, then the “New” button will be disabled. A geometry component is deleted by clicking on the “Delete” button. A 3D view of panels/components specified for the geometry can be accessed from the “View” button – see Section 1.2.4 for additional information about the 3D viewer.

When a geometry component is selected from the list of components, its corresponding geometry panels are displayed in the white text box on the right. Geometry panels can be added to the list of panels for a component by selecting one or more of the “Unused Panels” from the box on the left and clicking the “Add” button. Similarly, geometry panels belonging to a component can be



removed by selecting one or more panels from the box on the right and clicking the “Remove” button.



**Figure 16 Sample "Group Panels" Window**

This functionality was also provided and documented in VECC – see Section 2.4 of the VECC manual for additional information.

Note: Heating methods in HEAT-TK are based on simple geometry shapes. For example, vehicle nose cap is generally modeled using sphere heating method, wing leading edge is modeled using swept cylinder and flat acreage area is normally modeled either 2D wedge or cone method. Panels should be grouped according to the analysis method used.

### 3 Streamline Analysis (Quadstream)

HEAT-TK can be used to generate streamlines for a vehicle geometry, similar to the functionality provided in VECC. Quadstream is used to perform the streamline calculations. The streamline generation capability is accessed from the main window by clicking on the “Streamlines” button. This section provides an outline of the process of generating streamlines using HEAT-TK; for more detailed information about streamline parameters, see Section 3 of the VECC Software User’s Manual.

There are nine steps in the generate streamlines process: “Load Files”, “Set Panel Types”, “Group Panels”, “Specify Parts”, “Set Controls”, “Set Start/Stop”, “Save Inputs”, “Run Quadstream”, and “View Streamlines”, documented in the following sections.

#### 3.1 Load Files

The first step in the streamlines process is to specify the files for analysis. Figure 17 shows a sample window for the “Load Files” step. After specifying the name of an input file and the desired name for the results file, click the “Load Files” button to load the data into the process. Additional steps will then be accessible, with the number of accessible steps depending on how much data is available (e.g., a “.mk5” file contains the geometry specification along with parameter values related to generating streamlines, whereas a “.geo” file only contains the geometry specification).

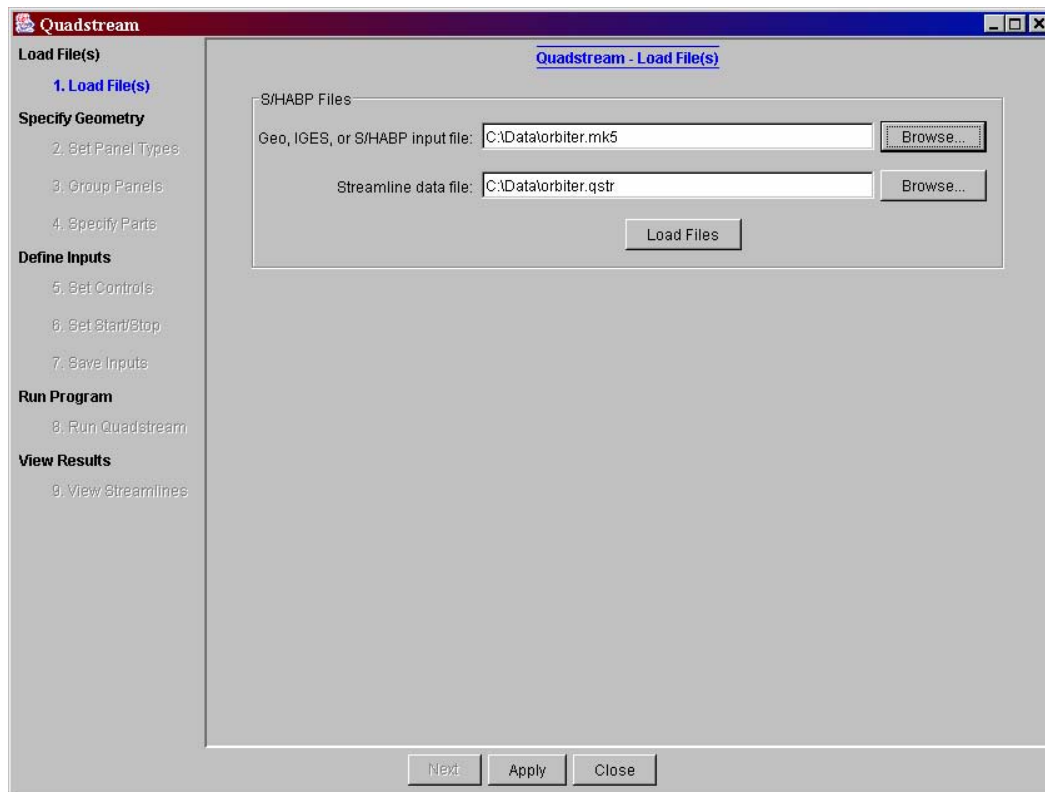


Figure 17 Streamlines - Load Files

### **3.2 Set Panel Types**

This step is common to several analysis processes and its functionality is documented in Section 2.2.

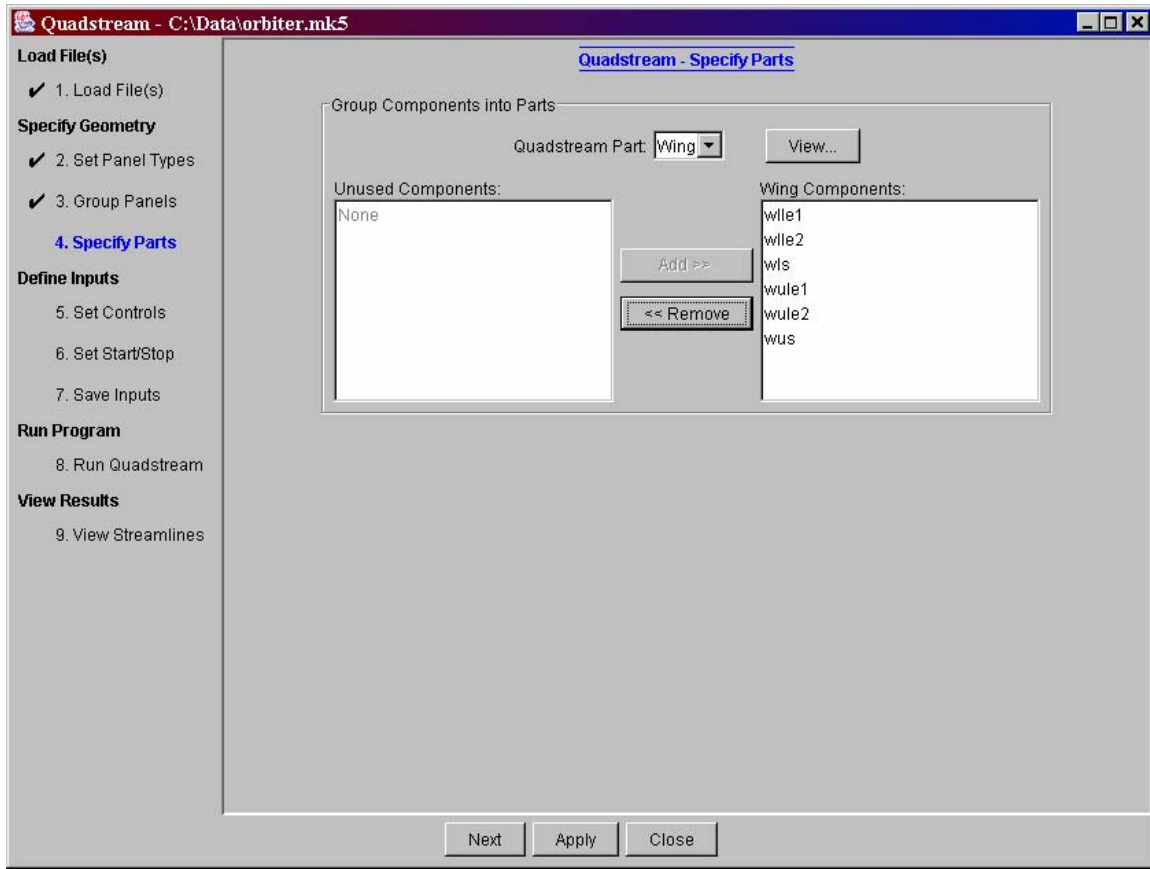
### **3.3 Group Panels**

This step is common to several analysis processes and its functionality is documented in Section 2.3.

### **3.4 Specify Parts**

Step 4 in the streamlines process groups S/HABP components into Quadstream parts. Figure 18 shows a sample window for the “Specify Parts” step. The standard Quadstream parts are given in the pull down list. When a Quadstream part is selected from the list, its corresponding geometry components are displayed in the white text box on the right. Geometry components can be added to the list of components for a part by selecting one or more of the “Unused Components” from the box on the left and clicking the “Add” button. Similarly, geometry components belonging to a part can be removed by selecting one or more components from the box on the right and clicking the “Remove” button. See Section 3.1.1 of the VECC Software User’s Manual for additional information.

A 3D view of components specified for the geometry can be accessed from the “View” button – see Section 1.2.4 for additional information about the 3D viewer.



**Figure 18 Streamlines - Specify Parts**

### 3.5 Set Controls

Step 5 in the streamlines process specifies parameters used to control various aspects of the Quadstream program, along with flight conditions. Figure 19 shows a sample window for the “Set Controls” step. See Sections 3.1.2 and 3.1.3 of the VECC Software User’s Manual for more information about these parameters.

**Note:** a new set of (alpha, beta) angles can be added in the flight conditions table by selecting the string “New” and overwriting it with a desired value. Make sure the <Enter> key is pressed on the keyboard after values for both alpha and beta are entered. A set of (alpha, beta) angles can be removed from the table by clearing one of the values and pressing the <Enter> key. The user can specify a maximum of 20 sets of (alpha, beta) angles.

Note: For aeroheating analysis, the user is required to generate from one to multiple sets of streamlines. The number of streamline sets required will depend on the range of alpha

and beta in the analysis design space. The program will use the closest match streamline set to the alpha and beta specified in the trajectory data point. The quality of the analysis will depend on how close the alpha/beta set matches the alpha and beta specify in the trajectory file. The program is not setup to interpolate between streamlines sets.

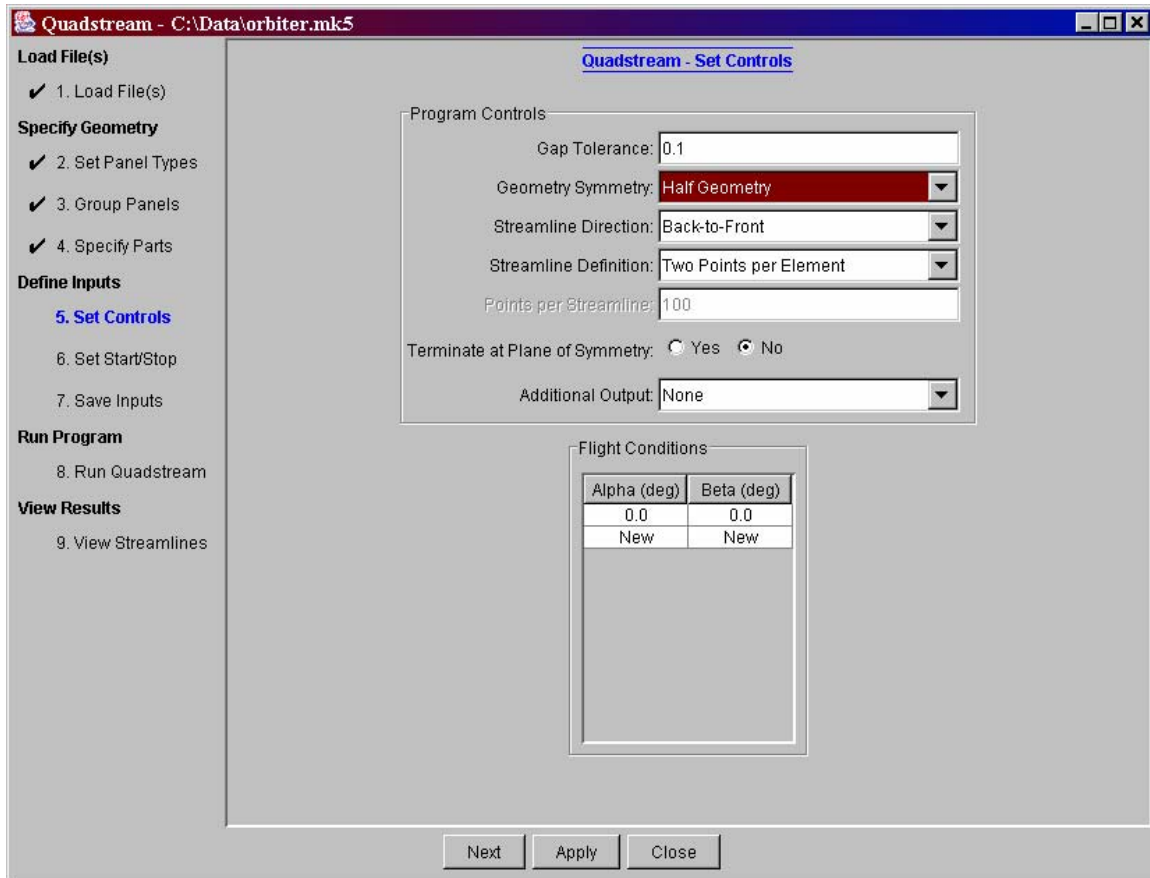
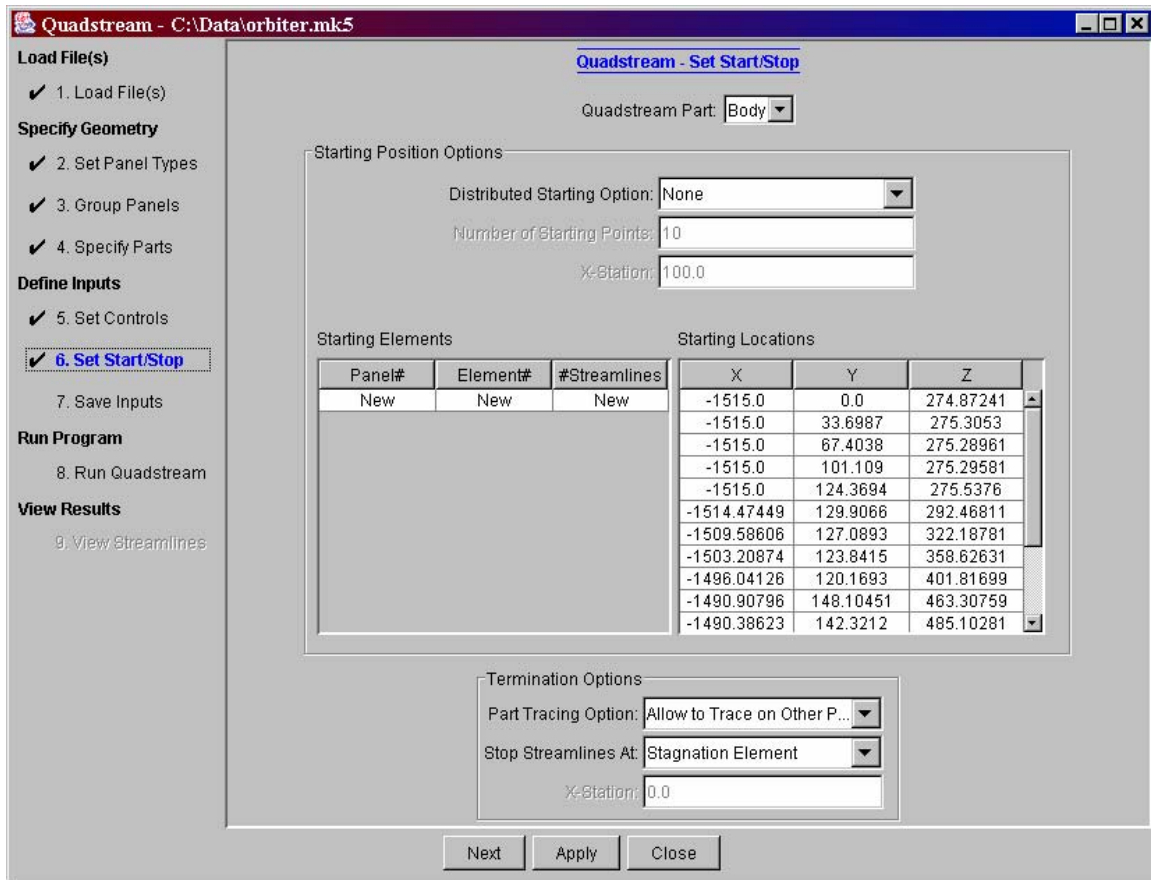


Figure 19 Streamlines - Set Controls

### 3.6 Set Start/Stop

Step 6 in the streamlines process specifies streamline starting and stopping conditions. Figure 20 shows a sample window for the “Set Start/Stop” step. See Section 3.1.4 of the VECC Software User’s Manual for more information about these parameters.

**Note:** a new row can be added in the “Starting Elements” table by selecting the string “New” and overwriting it with a desired value. Make sure the <Enter> key is pressed on the keyboard after the 3 values are entered. A row can be removed from the table by clearing one of the values and pressing the <Enter> key. A similar process is used for the “Starting Locations” table. To use these two options, the user has to select “None” for the distributed starting option.



**Figure 20 Streamlines - Set Start/Stop**

### **3.7 Save Inputs**

Step 7 in the streamlines process is used to save the input settings to a file. Figure 21 shows a sample window for the “Save Inputs” step. The “View Contents” button is used to view the contents of the file to be saved – after clicking on this button, the contents appear in the text area. The “Save” button is used to save the contents to a specified file.

**Note:** the values for all the input settings in the streamlines process can revert to the last saved values by going back to the “Load Files” step and clicking on the “Load Files” button.

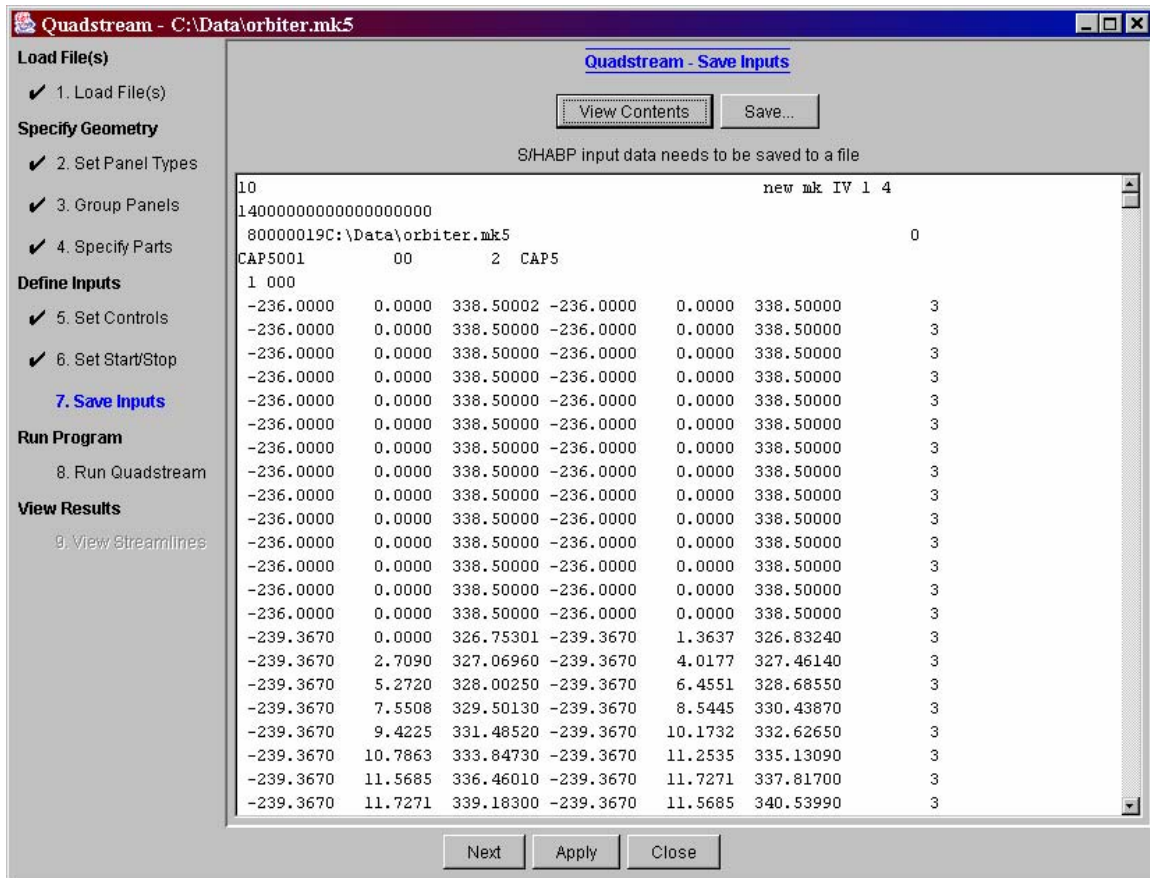


Figure 21 Streamlines - Save Inputs

### 3.8 Run Quadstream

Step 8 in the streamlines process is used to run Quadstream to generate streamlines. shows a sample window for the “Run Quadstream” step, as it appears after running the program. The “Run” button is used to run the program. An output file from running the program can be viewed by clicking on the “View Output” button and selecting a file. The contents of the selected file are displayed in the upper text area (unless the file is too large, in which case a separate window is used). The screen output from running the program is provided in the lower text area.

Running Quadstream produces an output file with the name specified in the “Load Files” step. A 3D view of the generated streamlines can be viewed in the succeeding step.

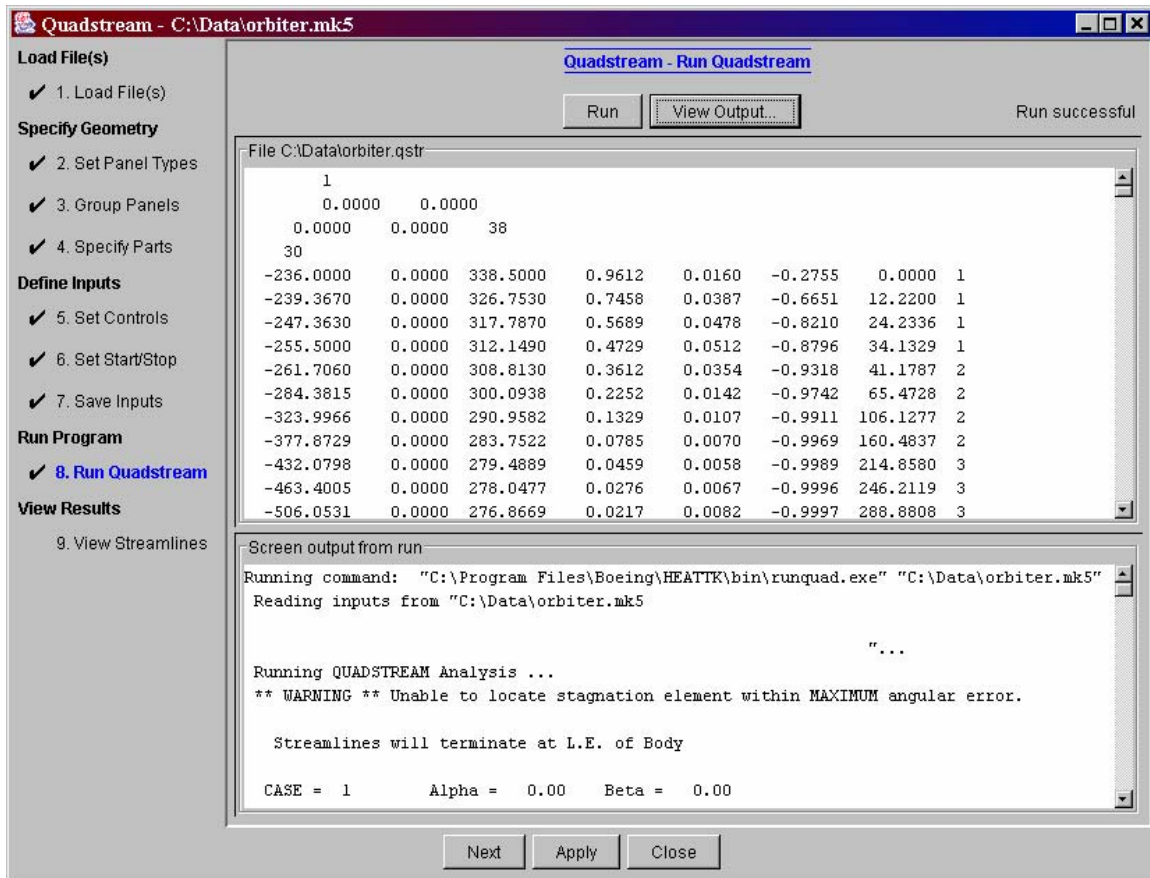


Figure 24 Streamlines - Run Quadstream

### 3.9 View Streamlines

The final step in the process is used to view streamlines generated by Quadstream. Figure 22 shows a sample window for viewing streamlines. Clicking on the “View” button brings up a separate browser window with a 3D view of the streamlines for the specified (alpha, beta) pair. Additional information about the controls appearing on the “View Streamlines” window and about the 3D viewer is available in Section 1.2.4.



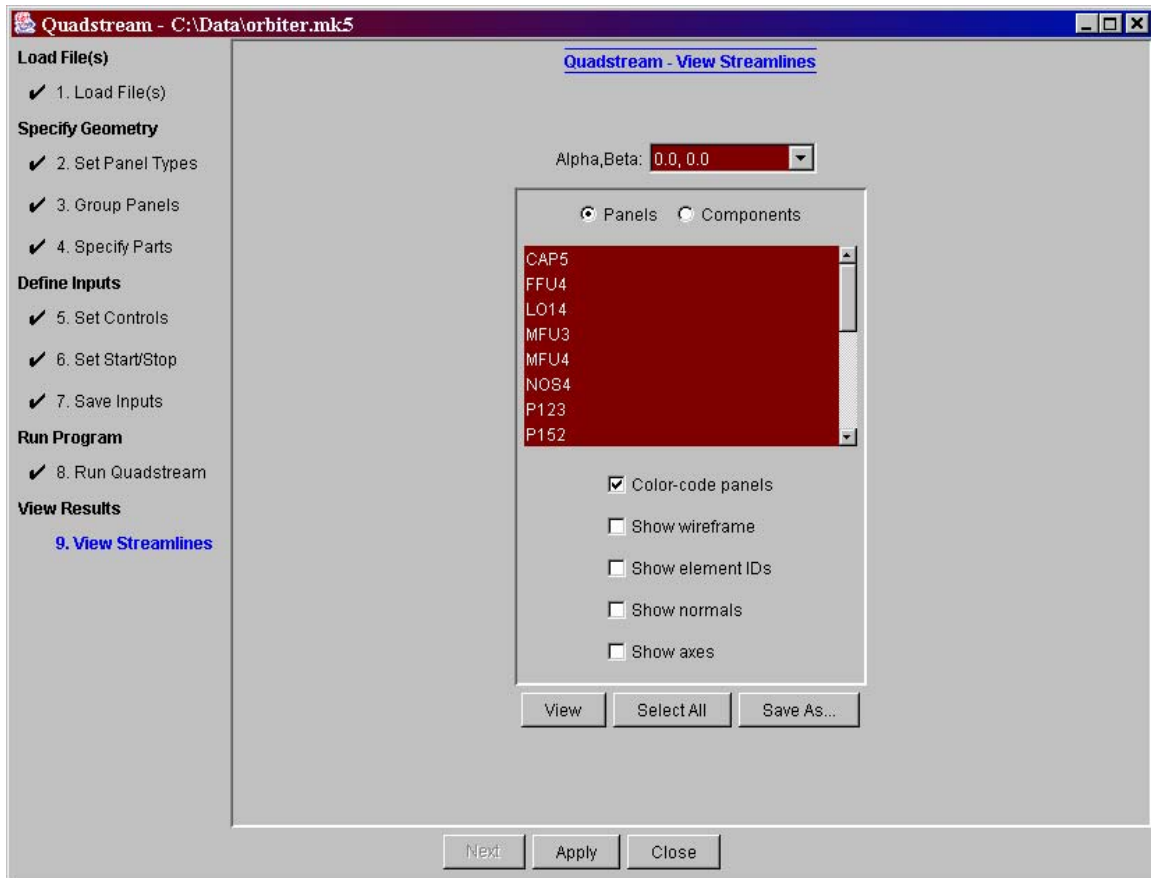


Figure 22 View Streamlines

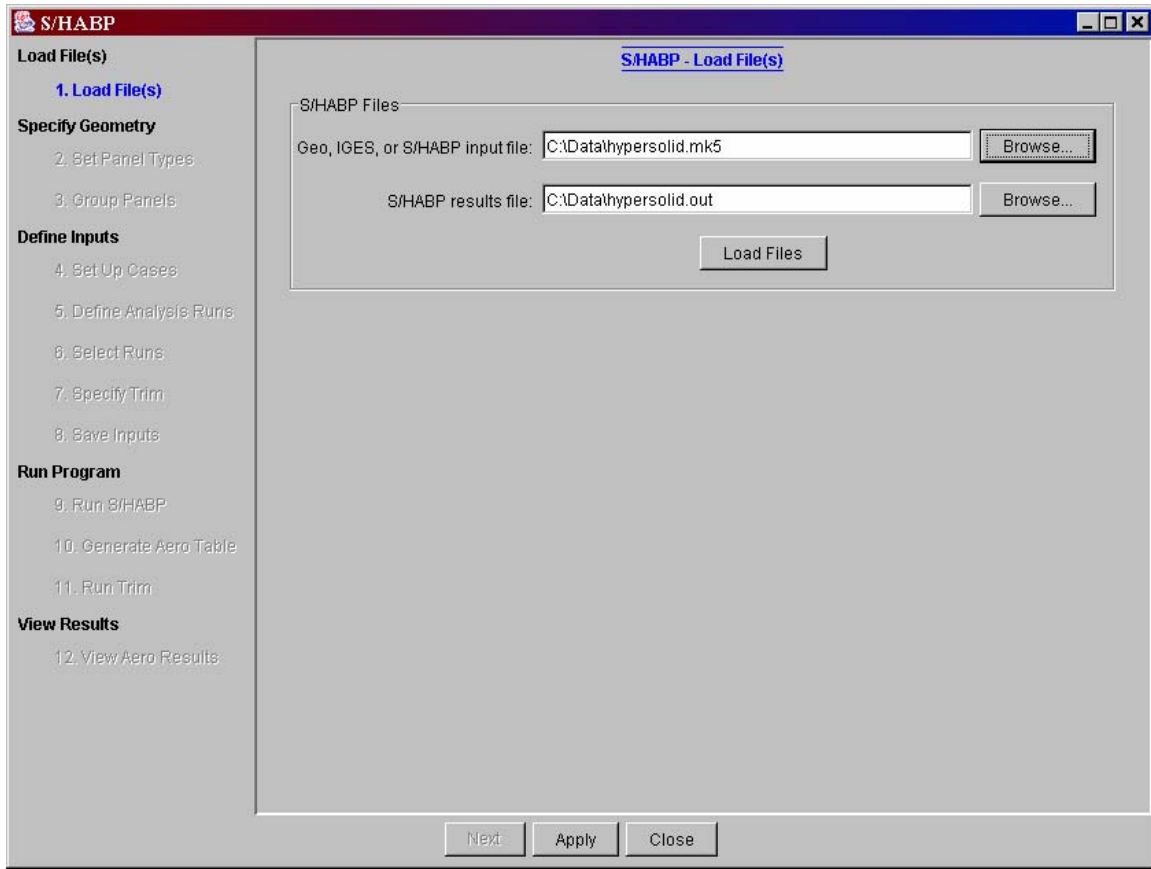
## 4 High Speed Aero Analysis (S/HABP)

HEAT-TK can be used to perform high-speed aero analysis on a vehicle geometry, similar to the functionality provided in VECC. The aero analysis results can optionally be trimmed during this process. S/HABP is used to perform the aero calculations; a separate trim program is used to trim the results. The high-speed aero/trim capability is accessed from the main window by clicking on the “HighSpeed Aero” button. This section provides an outline of the process of performing high-speed aero and trim analysis using HEAT-TK. For more detailed information about S/HABP input parameters, see Section 4 of the VECC Software User’s Manual. For information about trim input parameters, see Section 5 of the VECC manual.

There are twelve steps in the high-speed aero/trim analysis process: “Load Files”, “Set Panel Types”, “Group Panels”, “Set Up Cases”, “Define Analysis Runs”, “Select Runs”, “Specify Trim”, “Save Inputs”, “Run S/HABP”, “Generate Aero Table”, “Run Trim”, and “View Aero Results”, documented in the following sections.

### 4.1 Load Files

The first step in the high-speed aero process is to specify the files for analysis. Figure 23 shows a sample window for the “Load Files” step. After specifying the name of an input file and the desired name for the results file, click the “Load Files” button to load the data into the process. Additional steps will then be accessible, with the number of accessible steps depending on how much data is available (e.g., a “.mk5” file contains the geometry specification along with parameter values related to generating streamlines, whereas a “.geo” file only contains the geometry specification).



**Figure 23 High-Speed Aero - Load Files**

## **4.2 Set Panel Types**

This step is common to several analysis processes and its functionality is documented in Section 2.2.

## **4.3 Group Panels**

This step is common to several analysis processes and its functionality is documented in Section 2.3.

## **4.4 Set Up Trajectory**

Step 4 in the high-speed aero process specifies several input parameters related to S/HABP 'cases'. Figure 24 shows a sample window for the "Set Up Cases" step. See

Sections 4.1.1 and 4.1.4 of the VECC Software User's Manual for more information about these parameters.

A new case can be added for analysis by clicking the “New” button. This brings up a separate window for specifying a case name. Default values are used for its associated parameter settings. A copy of an existing case (with its associated parameter settings) can be added by selecting a case from the list of cases and clicking on the “Copy” button. After specifying a case name, the copied case is added. Finally, a case can be removed from the list by selecting a case and clicking on the “Delete” button.

**Note:** a new value can be added in the flight attitude table by selecting the string “New” from the table and overwriting it with a desired value. Make sure the <Enter> key is pressed on the keyboard after a value is entered. A value can be removed from the table by clearing one of the values and pressing the <Enter> key.

**S/HABP - Set Up Cases**

Case: Case 1 [New...] [Copy...] [Delete]

**Flight Conditions**

Conditions Input Mode: Standard Atmosphere

Gas Properties: Air

Mach Number: 5.0

Altitude: 0.0

Stagnation Pressure: 1.0

Stagnation Temperature: 0.0

**Reference Quantities**

Area: 10.0

Length: 100.0

Span: 10.0

Xcg: -50.0

Ycg: 0.0

Zcg: 0.0

**Flight Attitude**

Vary: Alpha

Constants	
Alpha	0.0
Beta	0.0
Roll	0.0
Pitch Rate	0.0
Yaw Rate	0.0
Roll Rate	0.0

**Derivative & Print Options**

Derivatives: None

Angular Increment: 1.0

Print Comp Force: ☒ Yes ☐ No

Echo Inputs to Out: ☐ Yes ☒ No

[Next] [Apply] [Close]

Figure 24 High-Speed Aero - Set Up Cases

## 4.5 Define Analysis Runs

Step 5 in the high-speed aero process specifies input parameters related to S/HABP ‘runs’, including runs for inviscid analysis, level-one viscous analysis, and level-two viscous analysis. Figure 25 shows a sample window for the “Define Analysis Runs” step.

The top half of the window displays a summary table of S/HABP runs already set up for analysis. A run can be selected by clicking on the circle appearing in the first column of the table. The corresponding parameter settings associated with a selected run then appears in the lower half of the window. The parameters are separated into several groups, one for each tab. The parameter settings can be changed before selecting another run or proceeding to the next step in the process.

**Note:** The parameters in the lower half of the window are mostly the same as available in VECC – they do not include the options related to flowfield analysis. See Section 4.4 of the VECC Software User’s Manual for additional information about inviscid analysis parameters, Section 4.5 for level-one viscous parameters, and Section 4.5 for level-two viscous parameters.

An S/HABP run can be added to the summary table by clicking the “New” button. This brings up a separate window where some basic parameters related to a run can be specified. Default values are used for other parameter settings. A copy of a run and its associated parameter settings can be added to the table by selecting a run from the table and clicking on the “Copy” button. This adds an entry at the bottom of the table of the copied run. A run can be removed from the table by selecting a run from the table and clicking on the “Delete” button.

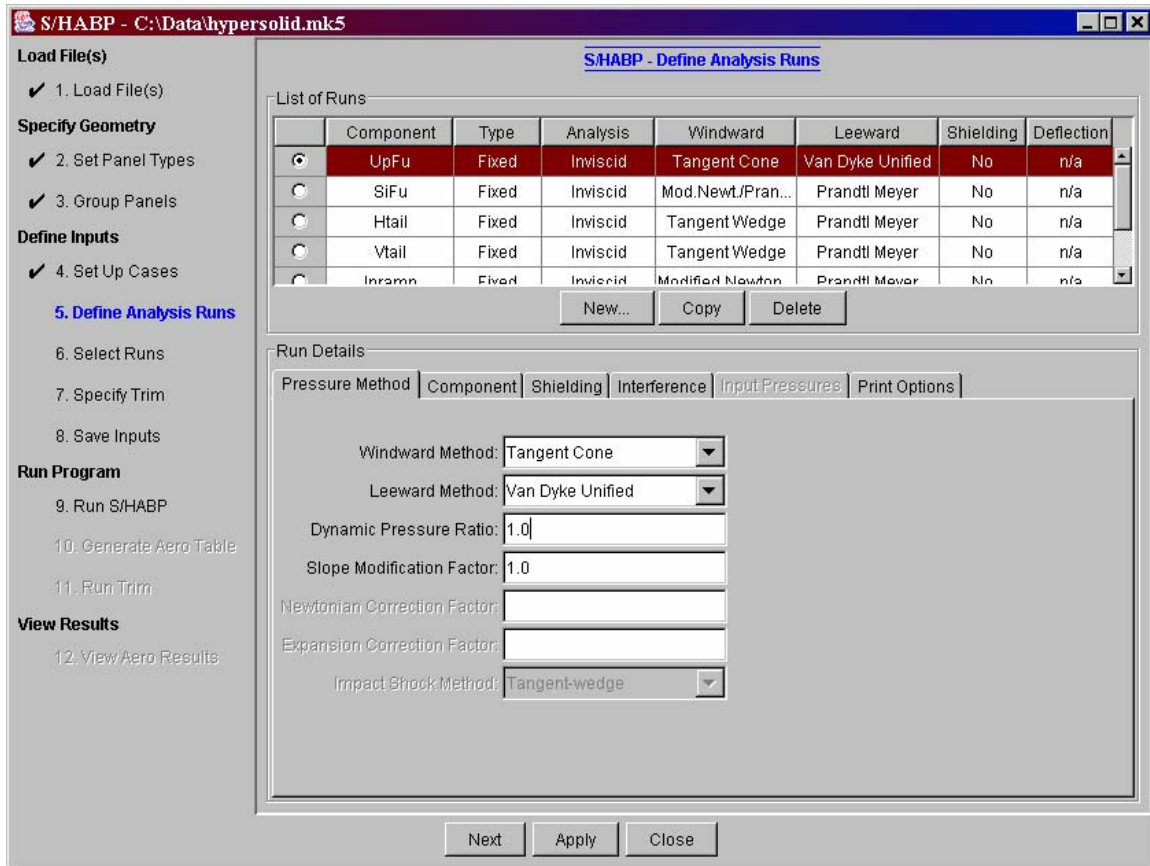


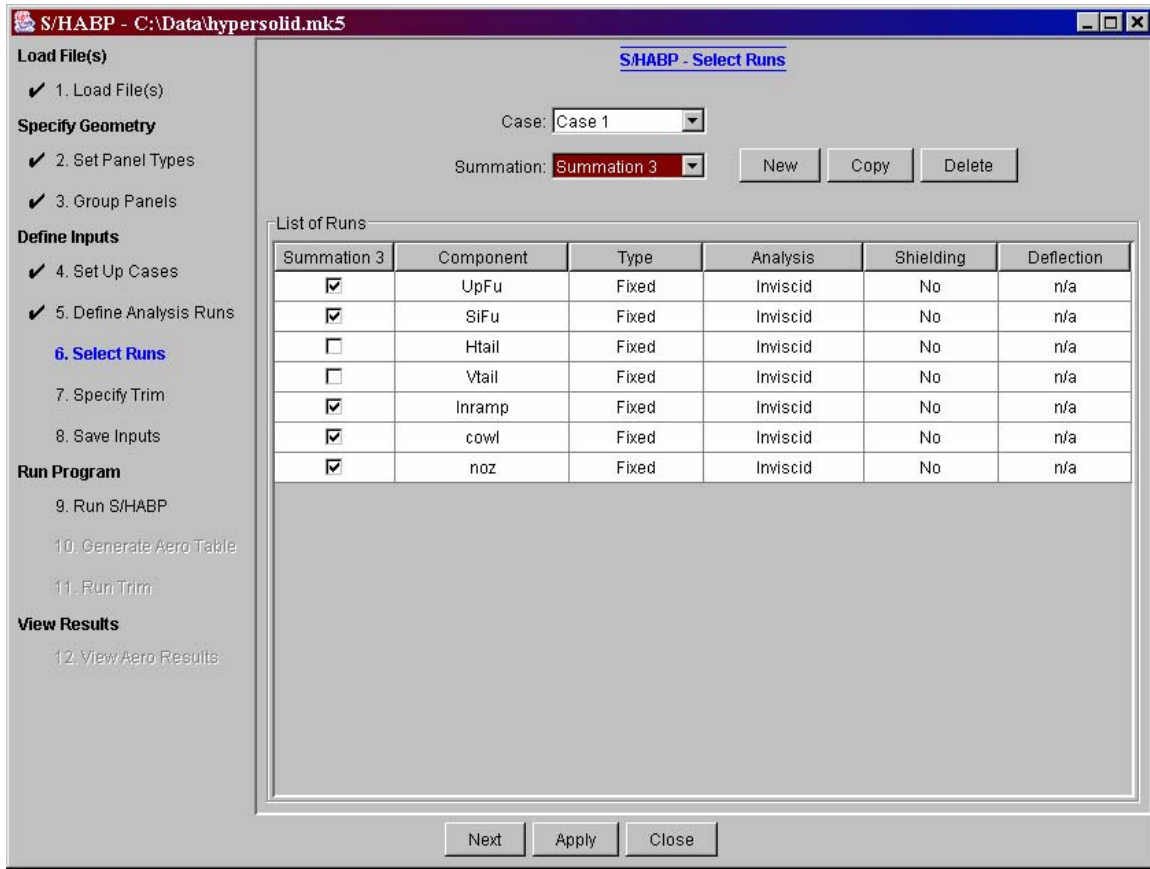
Figure 25 High-Speed Aero - Define Analysis Runs

## 4.6 Select Runs

Step 6 in the high-speed aero process specifies summations, used to sum the predictions from several runs. Figure 26 shows a sample window for the “Select Runs” step.

A new summation can be added to a case by selecting a case and clicking the “New” button. The new summation is added to the summation list, and it is numbered with the next sequential summation number for the case. Similarly, a copy of a summation and its selected runs can be added to a case by selecting a summation and clicking on the “Copy” button. Finally, a summation can be removed from the table by selecting a summation and clicking on the “Delete” button.

An S/HABP run can be added or removed from a summation via the checkbox in the table. Additional runs to be executed that are not included in any summation can be specified by selecting the “Non-summation run” field in the summation list and checking the runs to be executed.

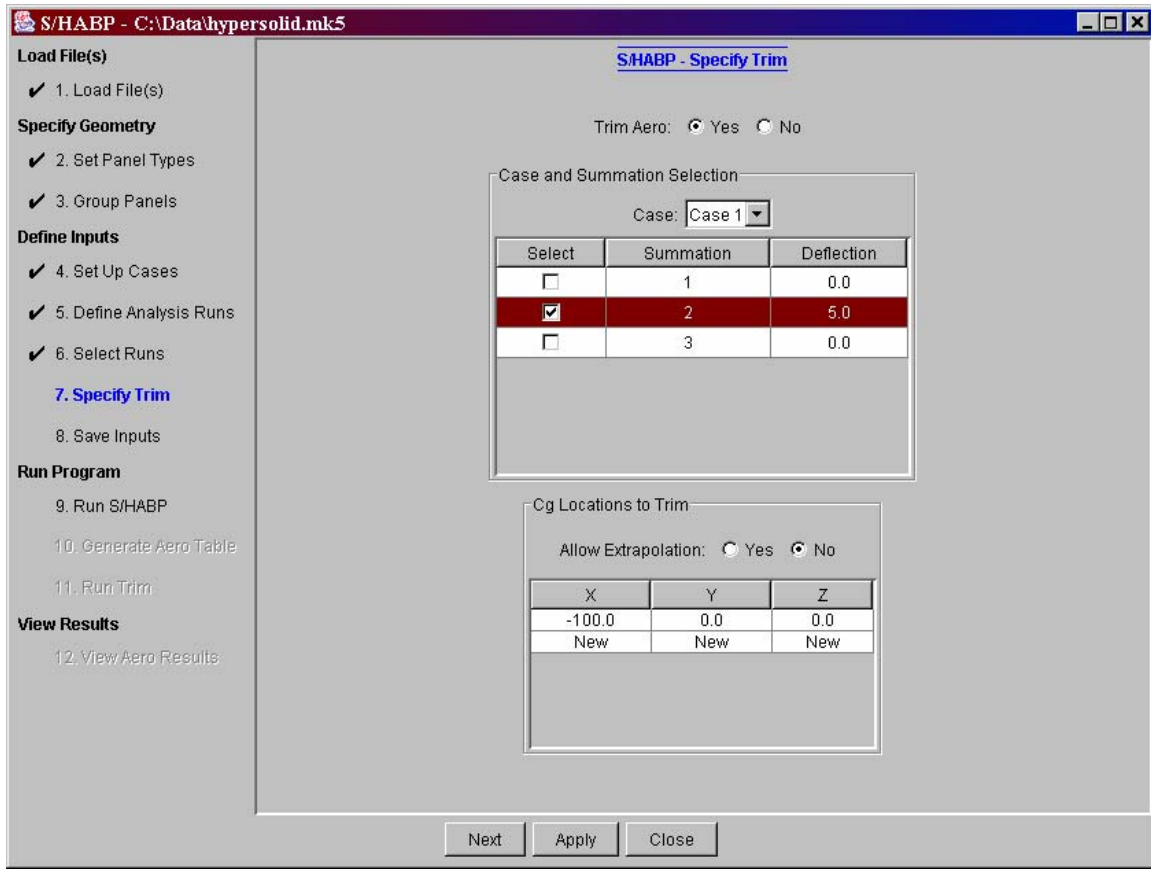


**Figure 26 High-Speed Aero - Select Runs**

## 4.7 Specify Trim

Step 7 in the high-speed aero process specifies whether the S/HABP aero results are to be trimmed. Figure 27 shows a sample window for the “Specify Trim” step. See Section 5.1 of the VECC Software User’s Manual for more information about these parameters.

**Note:** A summation can be selected using the checkbox in the “Select” column of the summation table. A deflection angle can be entered by overwriting the value in the “Deflection” column. Also, a new row can be added in the “Cg Locations” table by selecting the string “New” and overwriting it with a desired value. Make sure the <Enter> key is pressed on the keyboard after the 3 values are entered. A row can be removed from the table by clearing one of the values and pressing the <Enter> key.



**Figure 27 High-Speed Aero - Specify Trim**

## 4.8 Save Inputs

Step 8 in the high-speed aero process is used to save the input settings to a file. Figure 28 shows a sample window for the “Save Inputs” step. The “View Contents” button is used to view the contents of the file to be saved – after clicking on this button, the contents appear in the text area. The “Save” button is used to save the contents to a specified file.

**Note:** the values for all the input settings in the high-speed aero process can revert to the last saved values by going back to the “Load Files” step and clicking on the “Load Files” button.



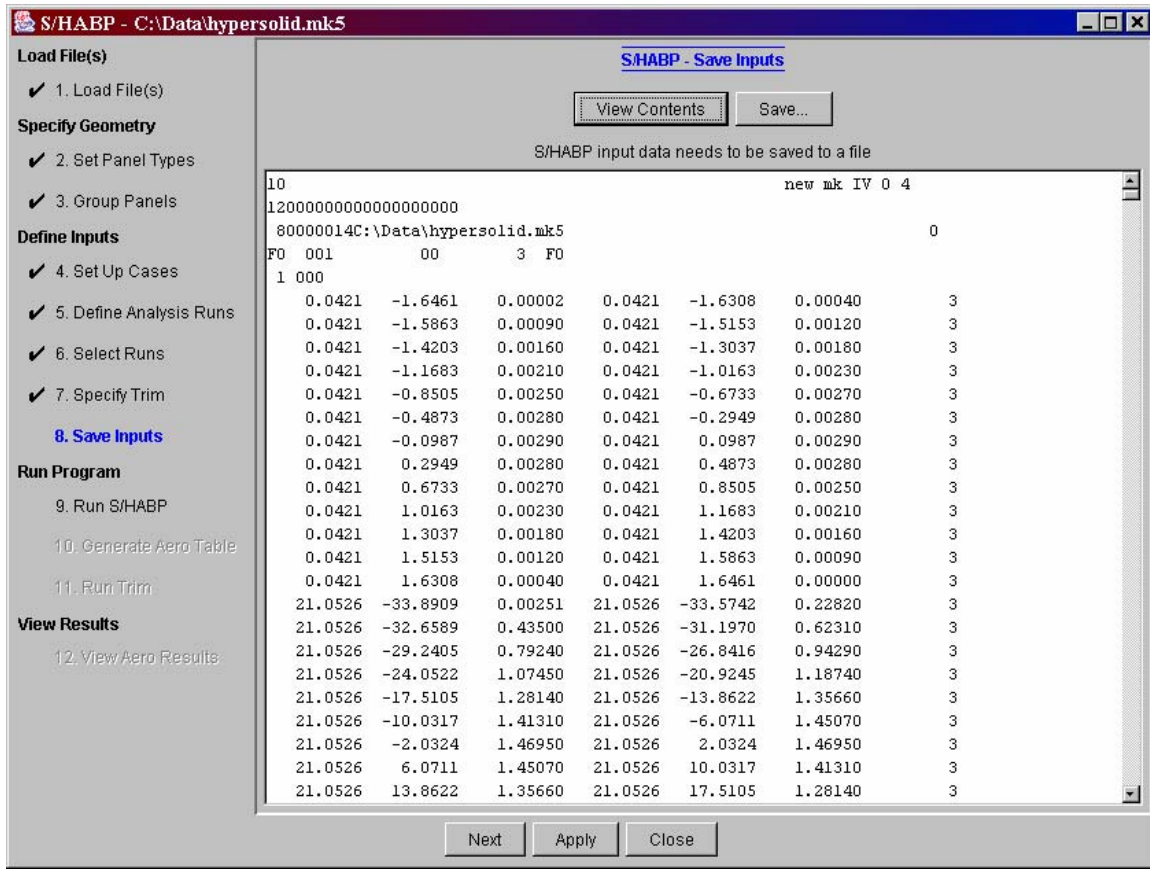
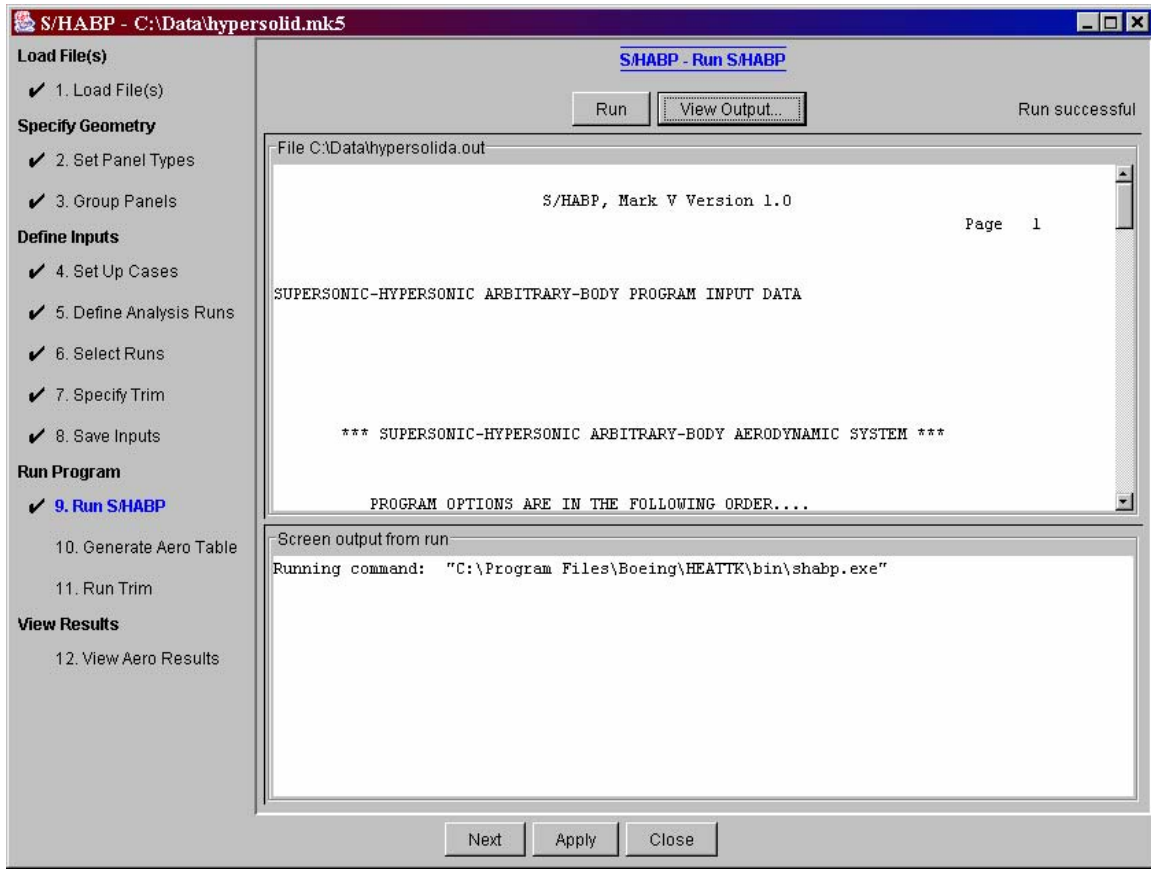


Figure 28 High-Speed Aero - Save Inputs

## 4.9 Run S/HABP

Step 9 in the high-speed aero process is used to run S/HABP to perform the specified analysis. Figure 29 shows a sample window for the “Run S/HABP” step, as it appears after running the program. The “Run” button is used to run the program. An output file from running the program can be viewed by clicking on the “View Output” button and selecting a file. The contents of the selected file are displayed in the upper text area (unless the file is too large, in which case a separate window is used). The screen output from running the program is provided in the lower text area.

Running S/HABP produces an output file with the name specified in the “Load Files” step. Graphical results can be viewed in a later step.



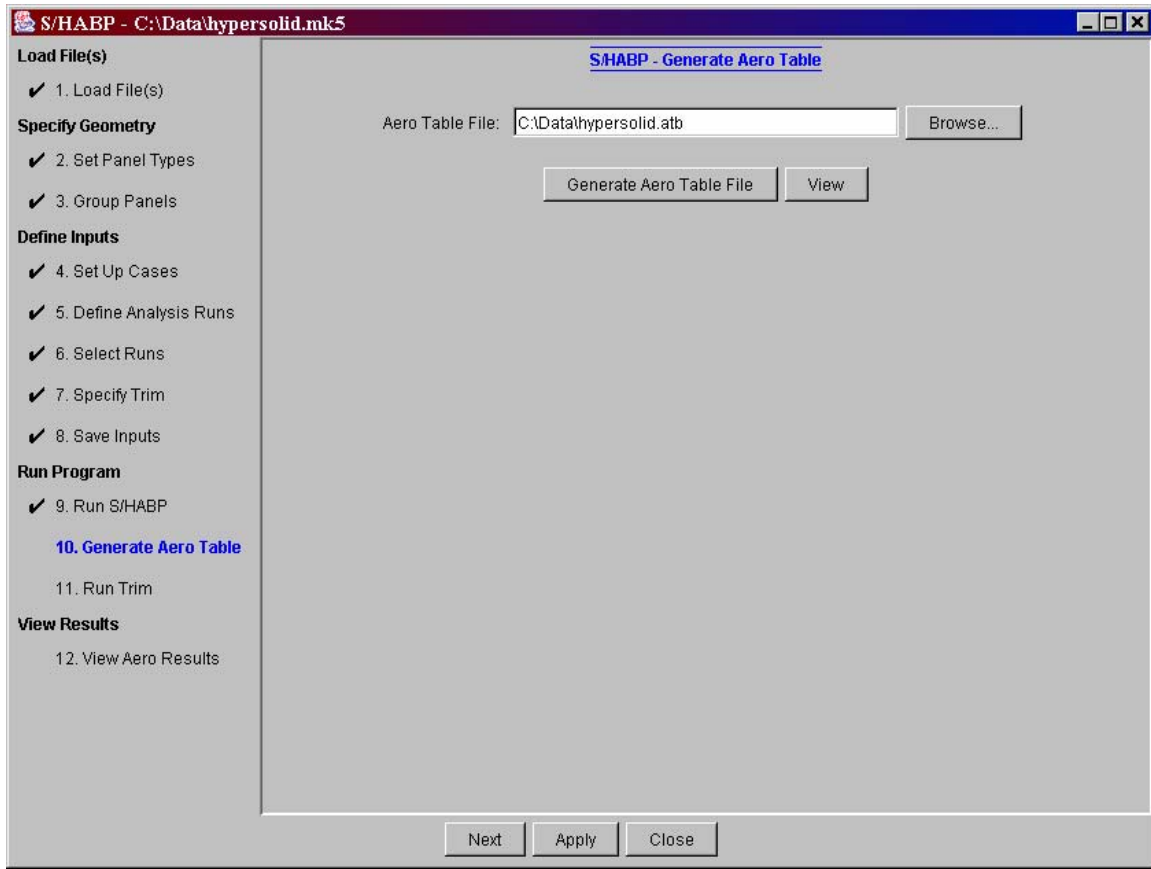
**Figure 29 High-Speed Aero - Run S/HABP**

## **4.10 Generate Aero Table**

Step 10 in the high-speed aero process is used to generate a HEAT-TK formatted aero table file from the S/HABP results. This only applies when:

1. there were inviscid runs having a pitch control component,
2. the force and moment print option was selected (step 5), and
3. the results are going to be used in a trajectory analysis using the Trajectory plug-in (documented in Section 6)

Figure 30 shows a sample window for the “Generate Aero Table” step. Specify an aero table file and then click on “Generate Aero Table File” to create the file. The resulting file can be viewed in a separate window by clicking the “View” button.



**Figure 30 High-speed Aero - Generate Aero Table**

#### **4.11 Run Trim**

Step 11 in the high-speed aero process is used to run a trim program to calculate the trimmed aerodynamics for the S/HABP results. Figure 31 shows a sample window for the “Run Trim” step, as it appears after running the program. The “Run” button is used to run the program. An output file from running the program can be viewed by clicking on the “View Output” button and selecting a file. The contents of the selected file are displayed in the upper text area (unless the file is too large, in which case a separate window is used). The screen output from running the program is provided in the lower text area.

Running the trim program produces an output file with a “.trm” suffix and the same prefix as the file specified in the “Load Files” step. Graphical results can be viewed in a later step.

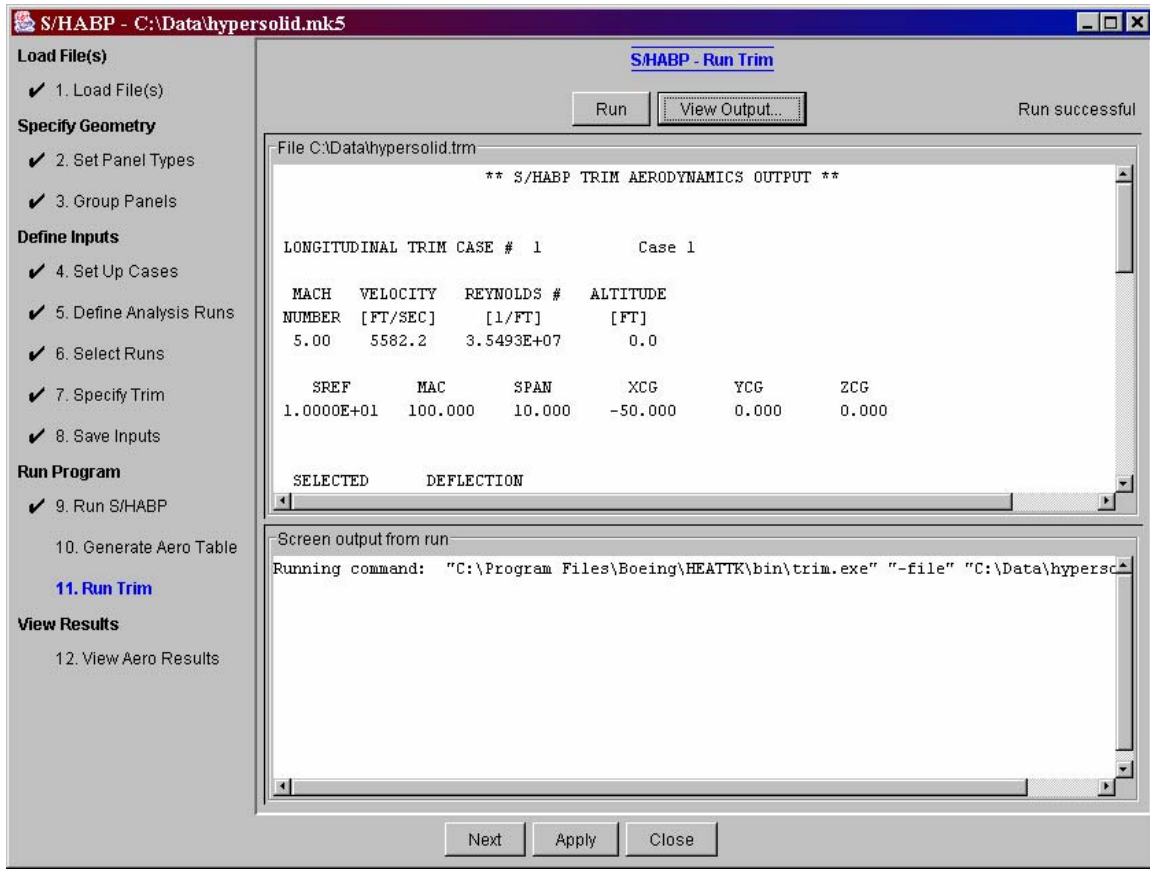


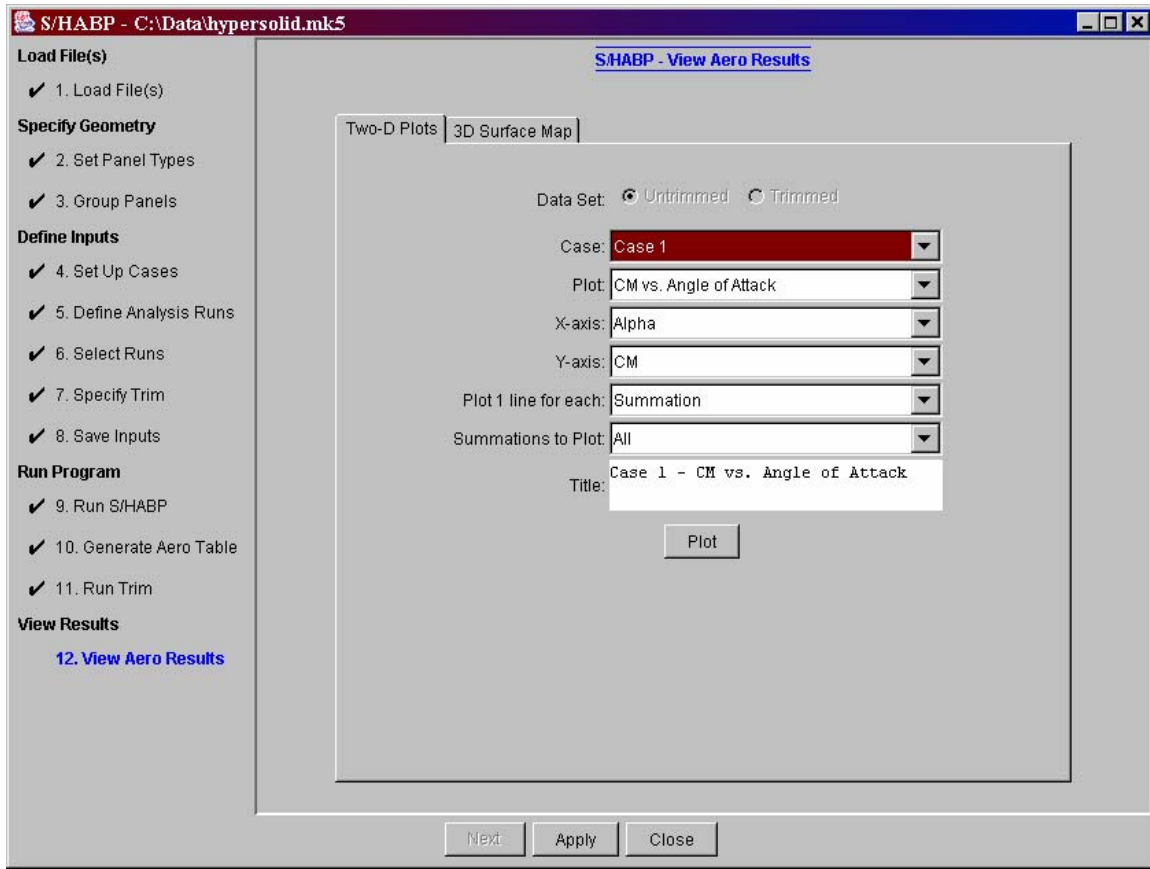
Figure 31 High-speed Aero - Run Trim

## 4.12 View Aero Results

The final step in the process is used to view graphical results from a high-speed aero analysis. The results can be viewed via 2D plots or a 3D surface map. The following sections outline these capabilities.

### 4.12.1 Two-D Plots

Two-D plots of the high-speed aero results are accessed by clicking on the “Two-D Plots” tab. Figure 32 shows a sample window for producing 2D plots. A plot is generated after selecting a data set (if option is available), a case, and what to plot. The “Plot” option specifies what to plot. If “Custom” is selected, then the x- and y-axis parameters can be specified. If one of the other options is selected (e.g., “CM vs. Angle of Attack”), the x- and y-axis parameters are set to correspond to the selection. Clicking on the “Plot” button brings up a separate window with the specified 2D plot. Additional information about the 2D plotter is available in Section 1.2.3.



**Figure 32 High-Speed Aero - 2D Plots**

#### 4.12.2 3D Surface Map

Three-D surface plots of the high-speed aero results are accessed by clicking on the “3D Surface Map” tab. Figure 33 shows a sample window for producing 3D surface map plots. A 3D surface view is specified by selecting a case, distribution values to display on the vehicle surface, and a flight condition (alpha, beta pair). The minimum and maximum values of the selected distribution are provided – the default is to generate a 3D view using this range of values. The minimum or maximum value to display can be adjusted. Clicking on the “View” button brings up a separate browser window with the requested 3D view. Additional information about the controls appearing on the right half of the 3D surface map window and about the 3D viewer is available in Section 1.2.4.

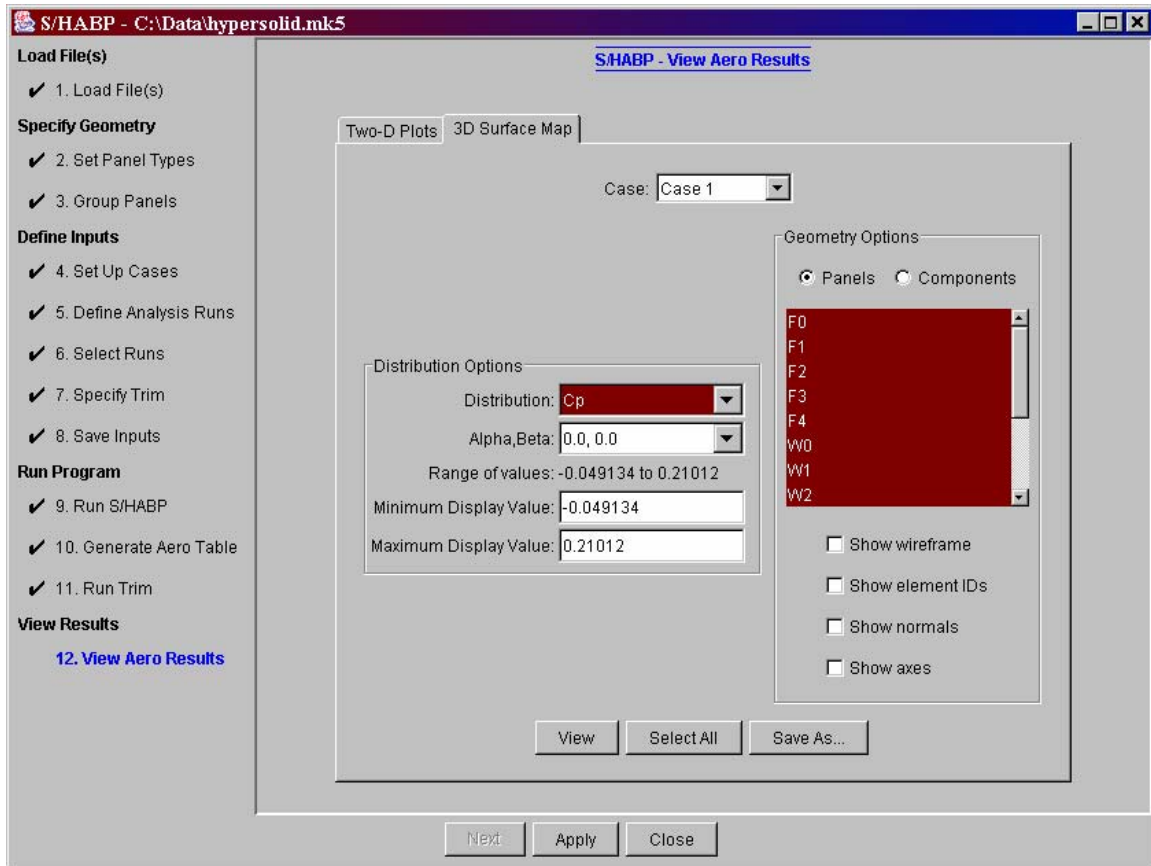


Figure 33 High-Speed Aero - 3D Surface Map

## 5 Low Speed Aero Plug-in

HEAT-TK provides an interface that allows a user-selected low-speed aerodynamic module to be easily plugged in. This low-speed aerodynamic plug-in is accessed from the main window by clicking on the “LowSpeed Aero” button. The plug-in has two steps in its process: “Setup Script” and “Run Script”, documented in the following sections. In order to use the results of the low-speed aero analysis in the trajectory plug-in, the data must be output in a standard format for a HEAT-TK aero table. Section 6.4 outlines the HEAT-TK aero table format.

**Note:** a sample script named ‘ssaero.bat’ is provided with the HEAT-TK sample data. This script runs a sample low-speed aero case using SSAERO and converts the results to the required format.

### 5.1 Setup Script

The first step in the low-speed aero plug-in is to setup information related to a script or executable program to run. Figure 34 shows a sample window for the “Setup Script” step.

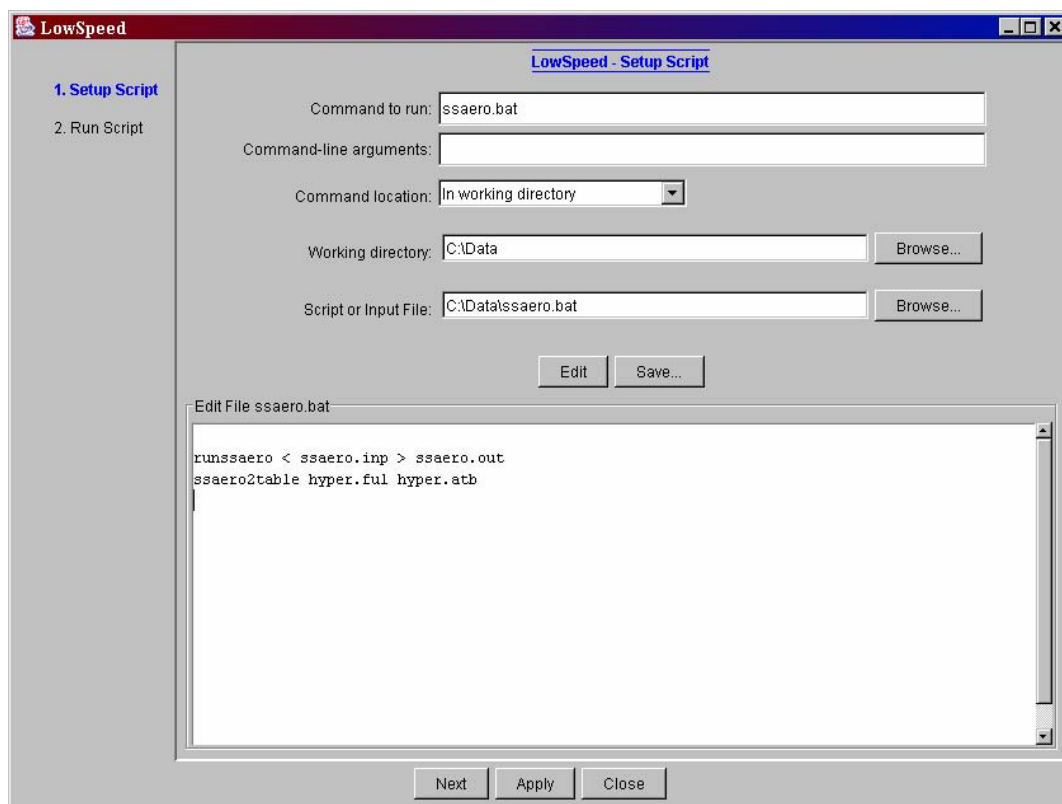


Figure 34 Low-Speed Setup Script

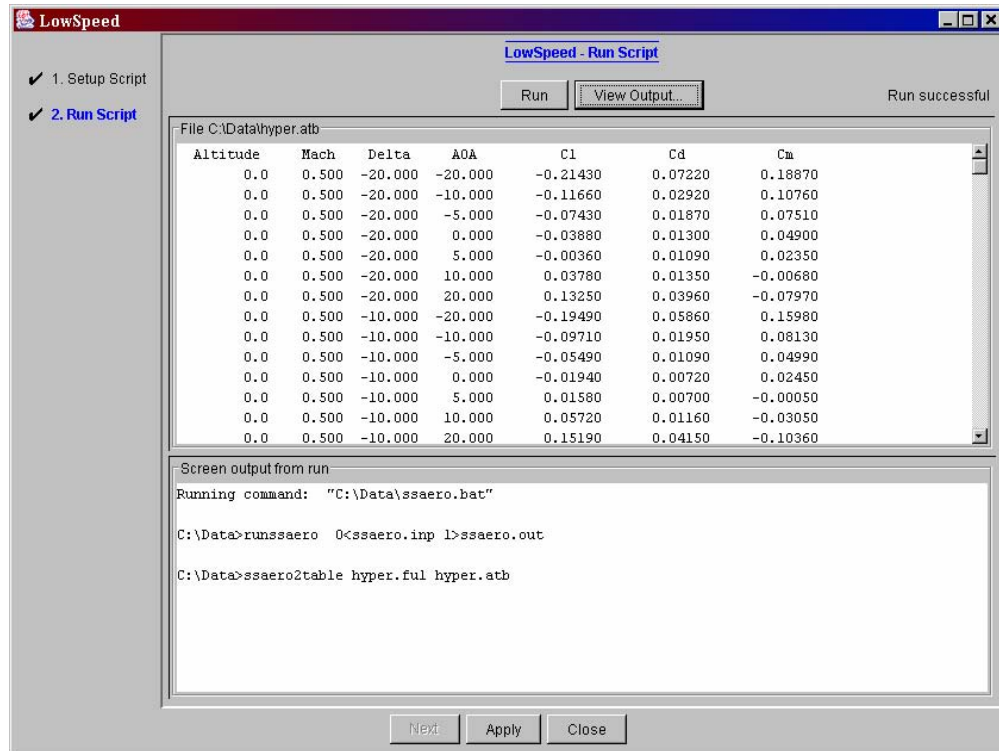
Here's a brief description of the setup script options:

- ❑ **Command to run** – specifies the command to use in running a script or executable for the low-speed analysis plug-in (this command will be executed in step two)
- ❑ **Command-line arguments** – specifies any command-line arguments to use in running the above command
- ❑ **Command location** – specifies where to find the command to run (either “In my path”, “In working directory”, or “Full path specified in command”)
- ❑ **Working directory** – specifies a directory where the command will be run from (and where data files are located)
- ❑ **Script or input file** – specifies the name of a file to edit in the white text area (only required if editing needs to be done or a file's contents is to be viewed). Click the “Edit” button to display this file in the text area. This file can be edited and then saved. Editing can also be done external to HEAT-TK.

## **5.2 Run Script**

The second step in the low-speed aero plug-in is to run the command that was specified in step one. Figure 35 shows a sample window for the “Run Script” step. The “Run” button is used to run the command. An output file from running the command can be viewed by clicking on the “View Output” button and selecting a file. The contents of the selected file are displayed in the upper text area (unless the file is too large, in which case a separate window is used). The screen output from running the command is provided in the lower text area.





**Figure 35 Low-Speed Run Script**

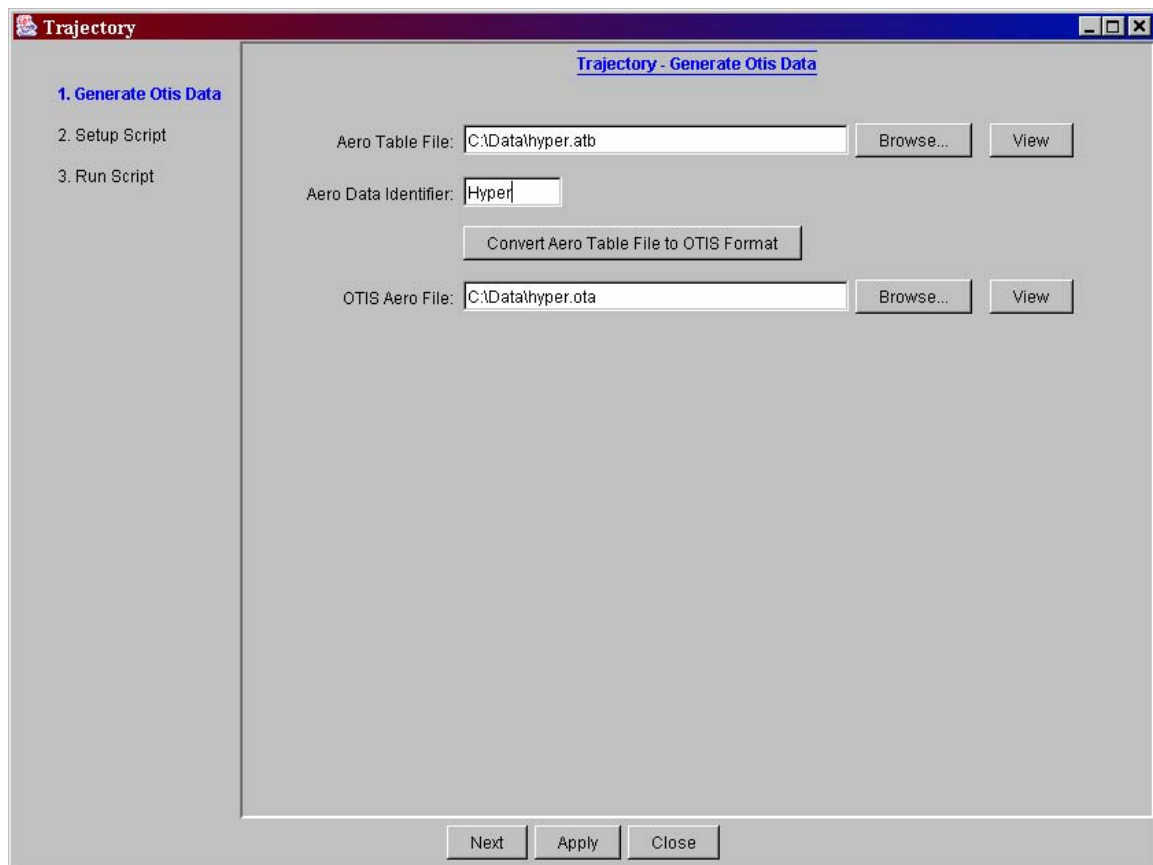
## 6 Trajectory Plug-in

HEAT-TK provides an interface that allows a user-selected trajectory analysis module to be easily plugged in. This trajectory plug-in is accessed from the main window by clicking on the “Trajectory” button. The plug-in has three steps in its process: “Generate Otis Data”, “Setup Script” and “Run Script”, documented in the following sections. After this, Section 6.4 outlines the required HEAT-TK data format for the results from aero analyses fed to the trajectory plug-in.

**Note:** a sample script named ‘map.bat’ is provided with the HEAT-TK sample data. This script runs a sample trajectory analysis using MAP.

### 6.1 Generate Otis Data

The first step in the trajectory plug-in is to convert data from the required HEAT-TK format into an alternate OTIS format. This is only required when using the MAP program supplied with HEAT-TK. Figure 36 shows a sample window for the “Generate Otis Data” step.



**Figure 36 Generate Otis Data for Trajectory Analysis**

A HEAT-TK aero table file requiring conversion is specified, along with an identifier (i.e., a label for the data; at most 6 characters). Clicking on the “Convert Aero Table File

to OTIS Format” button converts the given aero table file to the specified OTIS Aero File. This process can be repeated for each aero table file requiring conversion. Also, the “View” button can be used to display the contents of the corresponding file in a separate Notepad window.

## 6.2 Setup Script

The second step in the trajectory plug-in is to setup information related to a script or executable program to run. Figure 37 shows a sample window for the “Setup Script” step.

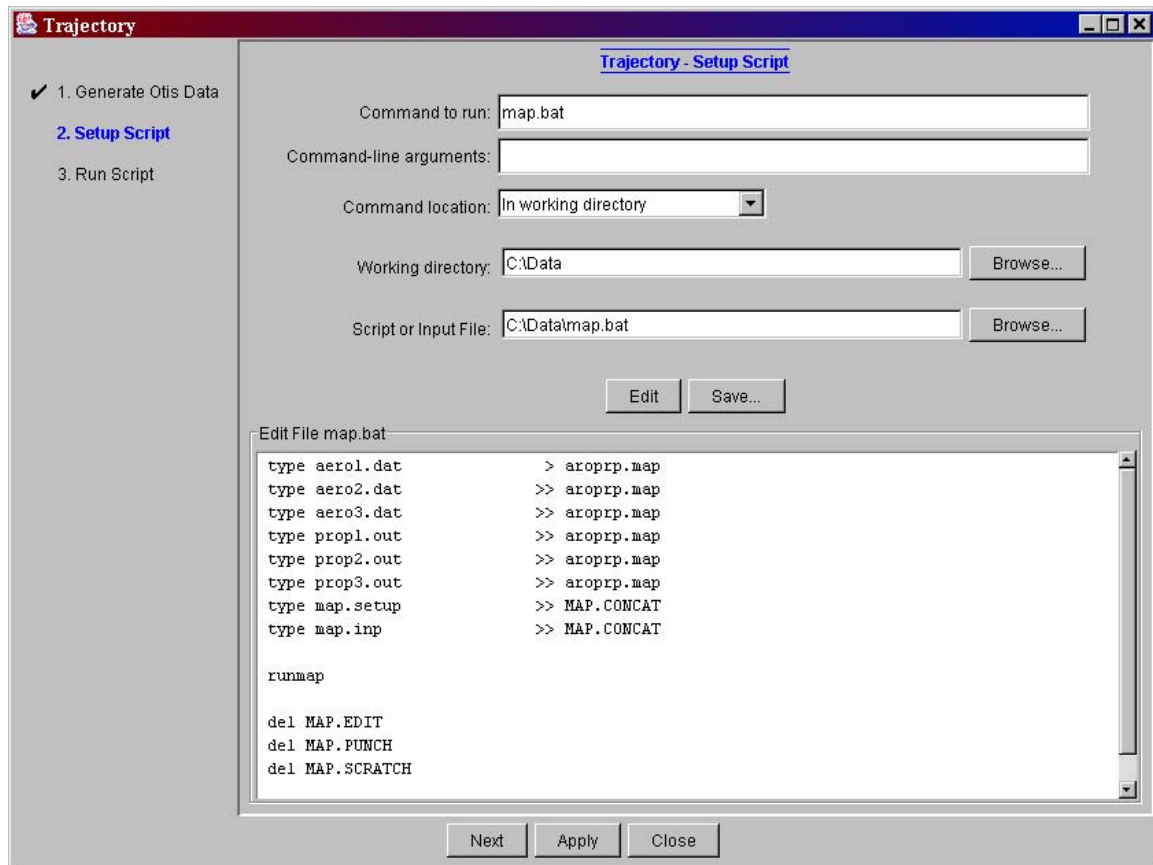


Figure 37 Trajectory Setup Script

Here’s a brief description of the setup script options:

- ❑ **Command to run** – specifies the command to use in running a script or executable for the trajectory analysis plug-in (this command will be executed in step three)
- ❑ **Command-line arguments** – specifies any command-line arguments to use in running the above command
- ❑ **Command location** – specifies where to find the command to run (either “In my path”, “In working directory”, or “Full path specified in command”)

- ❑ **Working directory** – specifies a directory where the command will be run from (and where data files are located)
- ❑ **Script or input file** – specifies the name of a file to edit in the white text area (only required if editing needs to be done or a file's contents is to be viewed). Click the “Edit” button to display this file in the text area. This file can be edited and then saved. Editing can also be done external to HEAT-TK.

### 6.3 Run Script

The third step in the trajectory plug-in is to run the command that was specified in step two. Figure shows a sample window for the “Run Script” step, as it appears after running the command. The “Run” button is used to run the command. An output file from running the command can be viewed by clicking on the “View Output” button and selecting a file. The contents of the selected file are displayed in the upper text area (unless the file is too large, in which case a separate window is used). The screen output from running the command is provided in the lower text area.

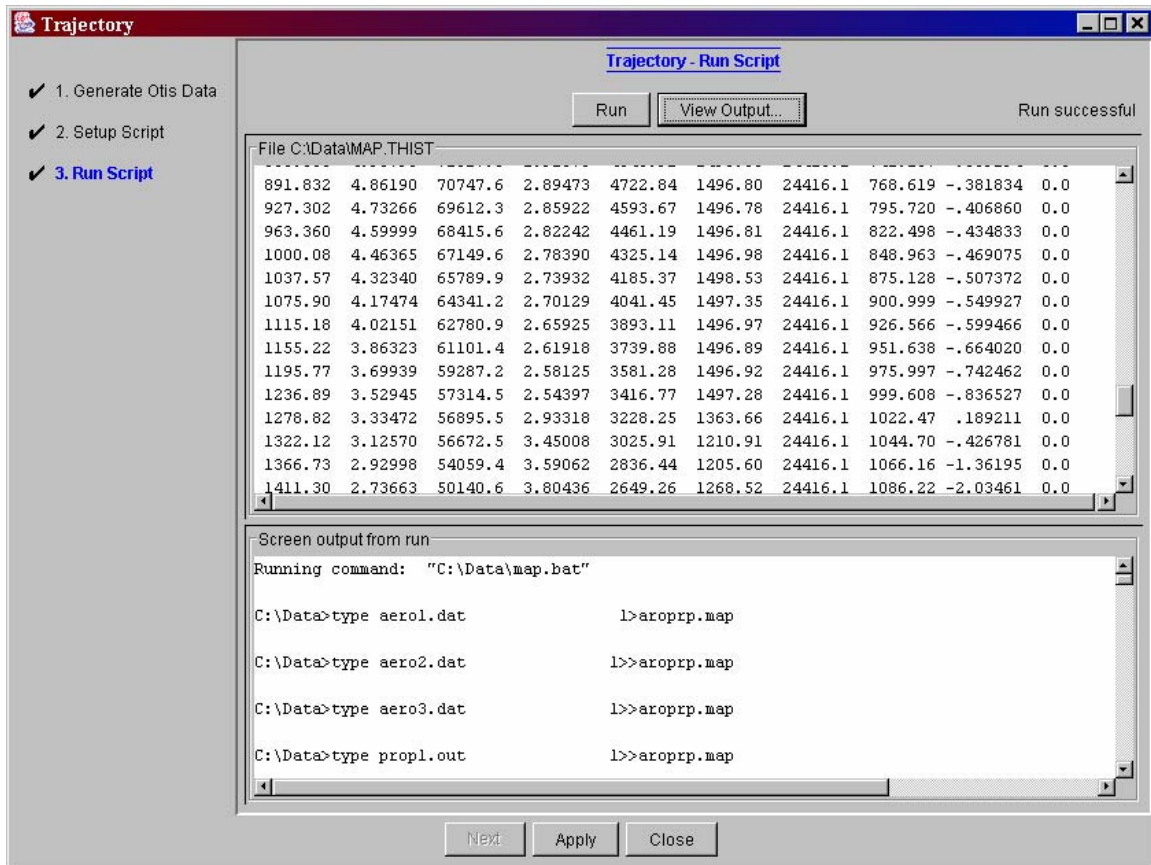


Figure 38 Trajectory Run Script

## 6.4 Aero Table Format

In order to use the results of the low-speed aero analysis in the trajectory plug-in, the data must be output in a standard format for HEAT-TK aero table. A sample of this data format is:

Altitude	Mach	Delta	AOA	C <sub>L</sub>	C <sub>D</sub>	C <sub>M</sub>
0.0	0.500	-20.000	-20.000	-0.21430	0.07220	0.18870
0.0	0.500	-20.000	-10.000	-0.11660	0.02920	0.10760
0.0	0.500	-20.000	-5.000	-0.07430	0.01870	0.07510
0.0	0.500	-20.000	0.000	-0.03880	0.01300	0.04900
0.0	0.500	-20.000	5.000	-0.00360	0.01090	0.02350
0.0	0.500	-20.000	10.000	0.03780	0.01350	-0.00680
0.0	0.500	-20.000	20.000	0.13250	0.03960	-0.07970
0.0	0.500	-10.000	-20.000	-0.19490	0.05860	0.15980
0.0	0.500	-10.000	-10.000	-0.09710	0.01950	0.08130
0.0	0.500	-10.000	-5.000	-0.05490	0.01090	0.04990
0.0	0.500	-10.000	0.000	-0.01940	0.00720	0.02450
0.0	0.500	-10.000	5.000	0.01580	0.00700	-0.00050
0.0	0.500	-10.000	10.000	0.05720	0.01160	-0.03050
0.0	0.500	-10.000	20.000	0.15190	0.04150	-0.10360
...						

The independent variables are altitude, mach, delta, and AOA. Note that the data is first grouped according to altitude; for a given altitude value, there are one or more groups of mach numbers. For a given mach number, there are one or more groups of trim surface deflection angles (delta). Finally, for a given delta angle, there are one or more angle of attack values (AOA). As illustrated above, this ordering results in the angle of attack values varying continuously. The data must be full-factorial in the independent variables. For a given altitude, mach, delta value, and angle of attack, the corresponding values for the dependent variables  $C_L$ ,  $C_D$ , and  $C_M$  values are specified.

## 7 Aeroheating Analysis (S/HABP)

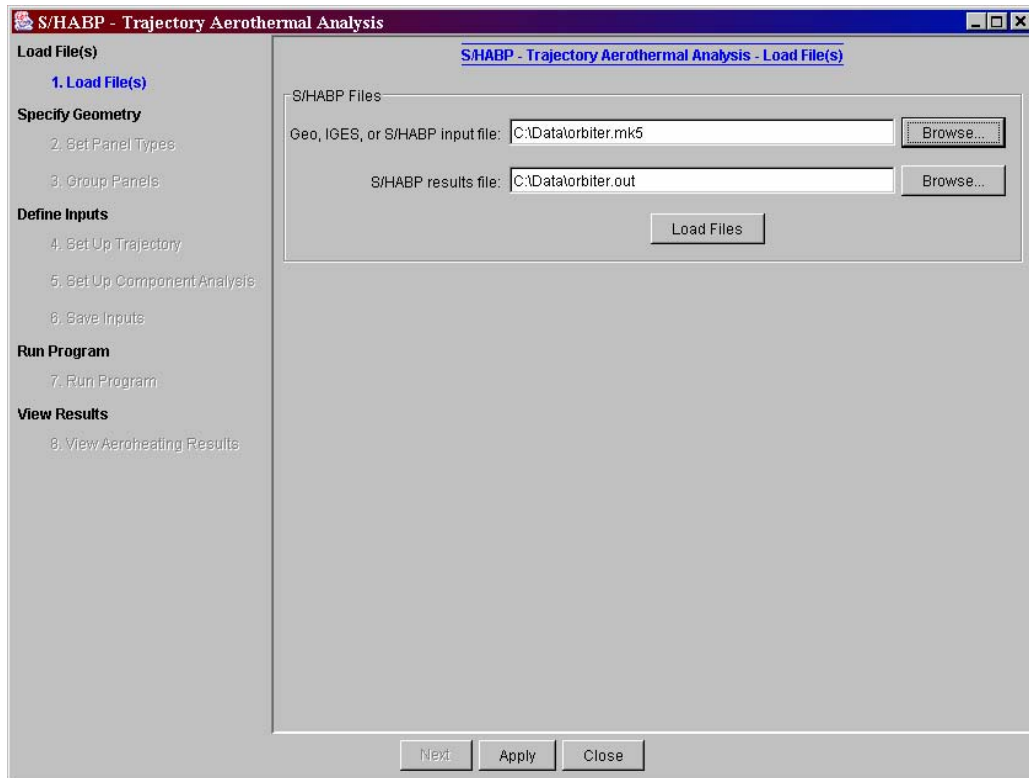
HEAT-TK can be used to perform Trajectory Aeroheating and viscous drag calculations along a mission trajectory profile. This trajectory aeroheating capability is accessed from the main window by clicking on the “Aeroheating” button. Streamlines must be generated for the vehicle geometry prior to doing trajectory aeroheating analysis; see Section 3 for documentation related to generating streamlines. S/HABP is used to perform the trajectory aeroheating calculations.

Note: For aeroheating analysis, the user is required to generate from one to multiple sets of streamlines. The number of streamline sets required will depend on the range of alpha and beta in the analysis design space. The program will use the closest match streamline set to the alpha and beta specified in the trajectory data point. The quality of the analysis will depend on how close the alpha/beta set matches the alpha and beta specify in the trajectory file. The program is not setup to interpolate between streamlines sets.

There are eight steps in the trajectory aeroheating process: “Load Files”, “Set Panel Types”, “Group Panels”, “Set Up Trajectory”, “Set Up Component Analysis”, “Save Inputs”, “Run Program”, and “View Aeroheating Results”, documented in the following sections.

### 7.1 Load Files

The first step in the trajectory aeroheating process is to specify the files for analysis. Figure 39 shows a sample window for the “Load Files” step. After specifying the name of an input file and the desired name for the results file, click the “Load Files” button to load the data into the process. Note that if a “.mk5” file is loaded, it must be formatted specifically for trajectory aeroheating analysis. Additional steps will then be accessible, with the number of accessible steps depending on how much data is available (e.g., a “.mk5” file contains the geometry specification along with parameter values related to trajectory aeroheating, whereas a “.geo” file only contains the geometry specification).



**Figure 39 Trajectory Aeroheating - Load Files**

## **7.2 Set Panel Types**

This step is common to several analysis processes and its functionality is documented in Section 2.2.

Note: For aeroheating analysis, the user is required to generate from one to multiple sets of streamlines. The number of streamline sets required will depend on the range of alpha and beta in the analysis design space. The program will use the closest match streamline set to the alpha and beta specified in the trajectory data point. The quality of the analysis will depend on how close the alpha/beta set matches the alpha and beta specify in the trajectory file. The program is not setup to interpolate between streamlines sets.

## **7.3 Group Panels**

This step is common to several analysis processes and its functionality is documented in Section 2.3.

Note: Heating methods in HEAT-TK are based on simple geometry shapes. For example, vehicle nose cap is generally modeled using sphere heating method, wing leading edge is modeled using swept cylinder and flat acreage area is normally modeled

either 2D wedge or cone method. Panels should be grouped according to the analysis method used.

## 7.4 Set Up Trajectory

Step 4 in the trajectory aeroheating process specifies several input parameters related to the trajectory analysis. Figure 40 shows a sample window for the “Set Up Trajectory” step.

S/HABP - Trajectory Aerothermal Analysis - C:\Data\orbiter.mk5

S/HABP - Trajectory Aerothermal Analysis - Set Up Trajectory

**Load File(s)**

- ✓ 1. Load File(s)

**Specify Geometry**

- ✓ 2. Set Panel Types
- ✓ 3. Group Panels

**Define Inputs**

- 4. Set Up Trajectory**
- 5. Set Up Component Analysis
- 6. Save Inputs

**Run Program**

- 7. Run Program

**View Results**

- 8. View Aeroheating Results

**Case Parameters**

Case Name: Case 1

Echo Inputs to Out: ☒ Yes ☐ No

Gas Properties: Air

Wall Condition: Constant Wall Temperature

Wall Temperature (1) 460.0

(2)

(3)

(4)

(5)

(6)

**Trajectory Parameters**

Trajectory File: C:\Data\STS2 Browse...

Start Time: 900.0

Time Step: 8.0

Stop Time: 1000.0

Next Apply Close

Figure 40 Trajectory Aeroheating - Set Up Trajectory

The following “Case Parameters” are specified in the top half of the window:

- ❑ **Case Name** – a name for this trajectory aeroheating case
- ❑ **Echo Inputs to Out** – whether to include the input settings in the results file
- ❑ **Gas Properties** – what gas properties to model (air or helium)



- ❑ **Wall Condition** – the wall boundary condition to use in the aeroheating analysis
- ❑ **Wall Temperature** – temperature values at which to perform the analysis; only valid when the wall condition is “Constant Wall Temperature”. Up to 6 temperature values can be entered, in degrees Rankine.

The following “Trajectory Parameters” are specified in the lower half of the window:

- ❑ **Trajectory File** – name and location of a file specifying trajectory information, such as altitude, speed, angle of attack, and control surface trim angle as functions of time.
- ❑ **Start Time** – starting time for the trajectory, in seconds
- ❑ **Time Step** – time slice to use in stepping along the trajectory, in seconds
- ❑ **Stop Time** – stopping time for the trajectory analysis, in seconds

Note: For TPS analysis, CHAP accepts no more than 200 time slices.

HEAT-TK accepts three types of trajectory files (Types 10, 70 and 80). Columns 1 and 2 of the first line of the trajectory file are always reserved for the trajectory type whereas the remaining columns are reserved for the title. Type 10 is the basic HEAT-TK trajectory file format (see Figure 41). The trajectory consists of time, altitude, freestream Mach number, angle of attack, sideslip angle, freestream pressure and temperature. If the freestream pressures set equal to zero in the trajectory file, the program will automatically use the standard atmospheric table. The last line of the Type 10 trajectory file ends with a “+” in Column 1. Type 70 trajectory file (see Figure 42), which is mainly used in the Space Shuttle mission type analysis, consists of time, altitude, freestream velocity, angle of attack, sideslip angle, two control surface deflection angles, freestream density and temperature. A user can also input type 70 trajectory data via free format by setting 70 to 7\* in the first line in the file. Similar to the Type 10 trajectory file, the last line of the trajectory data also ends with a “+” in Column 1. Type 80 trajectory file (see Figure 43), which is mainly used in the LANMIN aeroheating code, consists of time, altitude, velocity, angle of attack, sideslip angle, freestream density, temperature and pressure. Type 80 is also free format. The first column of the second line contains a “0” while the rest of the line is left blank. The last line of the trajectory file ends with a “1” in Column 1.

	Time(sec)	Alt.(ft)	Mach	Alpha(deg)	Beta(deg)	Poo (psf)	Too(deg R)
10	Case 1						
	0.000	100000.00	3.000	0.000	0.000	0.00	0.000
	1.000	90000.00	5.000	30.000	0.000	0.00	0.000
	2.000	100000.00	3.000	0.000	0.000	0.00	0.000

**Figure 41 Example of Type 10 Trajectory Data File**

Time(sec)	Alt.(ft)	Vel(fps)	Alpha(deg)	Beta(deg)	$\delta_e$ (deg)	$\delta_{br}$ (deg)	$\rho_{oo}$ (slug/ft <sup>3</sup> )	T <sub>oo</sub>
70	SHABP Test Trajectory							
	0.0	399829.324420.6	40.00	0.0	0.0	-4.403.6358D-11	715	
	8.00395725.824425.7		40.00	0.0	0.0	-4.404.1563D-11	648	
	16.00391633.224430.8		40.00	0.0	0.0	-4.404.7115D-11	595	
	24.00387551.824435.9		40.00	0.0	0.0	-4.405.5169D-11	569	
	32.00383482.124441.0		40.00	0.0	0.0	-4.406.5302D-11	546	
	.		.		.			
	.		.		.			
	.		.		.			
	1176.00162125.310119.9		38.62	0.0	-10.00	3.732.3321D-06	486	
	1184.00160618.0	9866.0	38.13	0.0	-10.00	3.812.4648D-06	486	
	1192.00159288.3	9616.4	37.78	0.0	-10.00	3.822.5881D-06	487	
	1200.00158064.1	9371.4	37.47	0.0	-10.00	3.882.7070D-06	487	
+								

Figure 42 Example of Type 70 Trajectory Data

Time(sec)	Alt.(ft)	Vel(fps)	Alpha(deg)	Beta(deg)	$\rho_{oo}$ (slug/ft <sup>3</sup> )	T <sub>oo</sub> (deg R)	P00(psf)
80,Type 80 File Format							
0,							
0,0.00,	399998.,	24195.5,	39.99,	0.00,	3.31E-11,	697.2,	4.41E-05,
0,50.00,	392728.,	24204.5,	39.99,	0.00,	4.27E-11,	650.2,	4.76E-05,
0,100.00,	385670.,	24213.1,	39.99,	0.00,	5.56E-11,	603.7,	5.76E-05,
0,150.00,	378939.,	24221.1,	39.99,	0.00,	7.36E-11,	559.2,	7.06E-05,
0,200.00,	372499.,	24228.6,	39.99,	0.00,	9.82E-11,	516.7,	8.71E-05,
0,250.00,	366367.,	24235.5,	39.99,	0.00,	1.32E-10,	476.2,	1.08E-04,
0,300.00,	360556.,	24241.6,	39.99,	0.00,	1.79E-10,	439.4,	1.35E-04,
0,350.00,	355080.,	24246.9,	39.99,	0.00,	2.42E-10,	409.6,	1.70E-04,
0,400.00,	349959.,	24251.2,	39.99,	0.00,	3.17E-10,	392.1,	2.14E-04,
0,450.00,	345263.,	24254.1,	39.99,	0.00,	4.07E-10,	380.4,	2.66E-04,
	.	.	.				
	.	.	.				
	.	.	.				
0,2900.00,	22400.,	417.2,	10.00,	0.00,	1.16E-03,	438.6,	8.73E+02,
0,2950.00,	16216.,	400.3,	10.00,	0.00,	1.43E-03,	460.7,	1.13E+03,
0,3000.00,	9913.,	462.0,	6.43,	0.00,	1.76E-03,	483.2,	1.46E+03,

Figure 43 Example of Type 80 Trajectory Data File

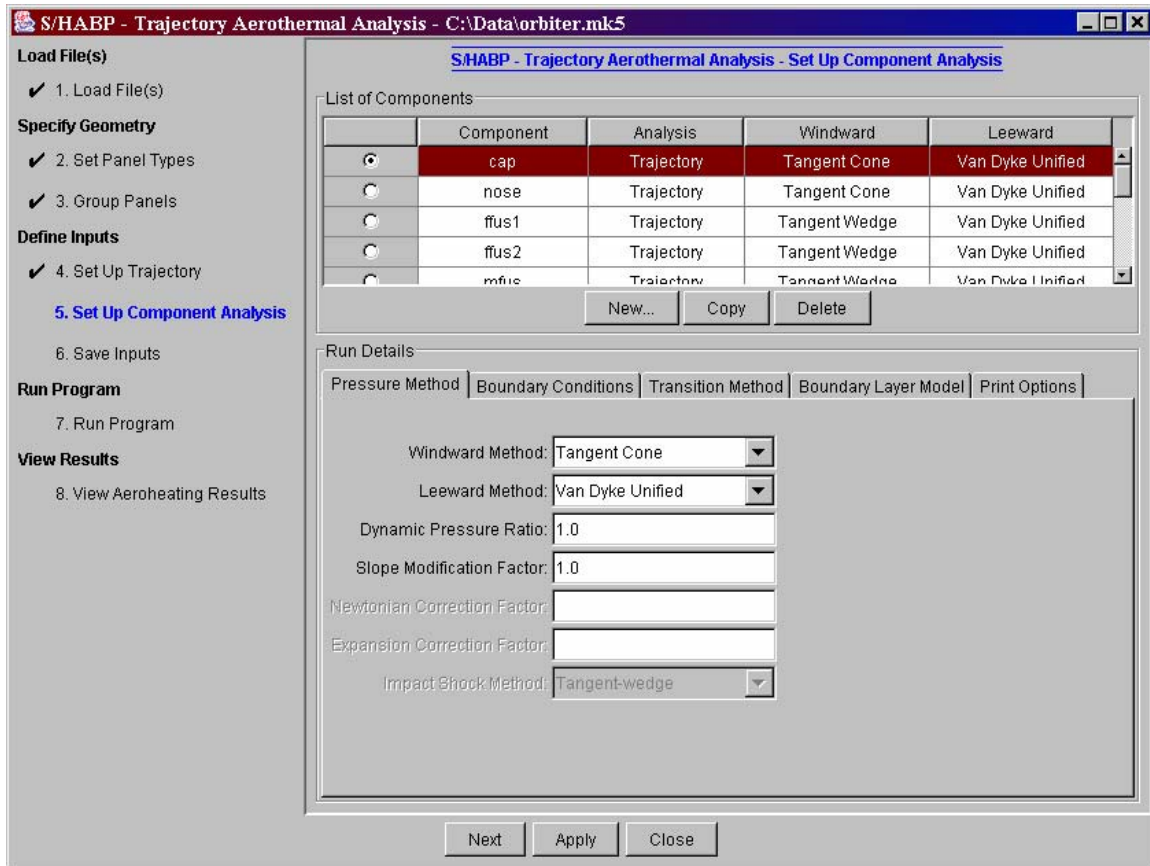
## 7.5 Set Up Component Analysis

Step 5 in the trajectory aeroheating process specifies input parameters related to each geometry component. Figure 44 shows a sample window for the “Set Up Component Analysis” step.

The top half of the window displays a summary table of geometry components already set up for analysis. A geometry component can be selected by clicking on the circle appearing in the first column of the table. The corresponding parameter settings associated with a selected geometry component then appears in the lower half of the window. The parameters are separated into 5 groups, one for each tab. The parameter settings can be changed before selecting another geometry component or proceeding to the next step in the process.

**Note:** The parameters in the lower half of the window are the same as available in VECC, for Level 2 Viscous analysis. See Section 4.6 of the VECC Software User’s Manual for additional information about these parameters.

A geometry component can be added to the summary table by clicking the “New” button. This brings up a separate window where a geometry component and its methods for windward and leeward analysis can be specified. Default values are used for its associated parameter settings. A copy of a geometry component and its associated parameter settings can be added to the table by selecting a geometry component from the table and clicking on the “Copy” button. This adds an entry at the bottom of the table of the copied geometry component. A geometry component can be removed from the table by selecting a geometry component from the table and clicking on the “Delete” button.



**Figure 44 Trajectory Aeroheating - Set Up Component Analysis**

**Note:** a new feature has been added to the radius of influence controls parameter to allow the user to specify the number of data points selected within the radius of influence for interpolation. To use this new feature the user must input a negative radius of influence (e.g. -30.010). The absolute value of the integer to the left hand side of the decimal point represents the radius of influence, in this case is equal to 30. The integer to the right hand side of the decimal point, in this case equal to 10, is used to specify the maximum number of data points to be used for interpolation. In this case, the program will choose the nearest 10 data points for interpolation. It should be noted that in the gas equation option selection, if the user chooses real gas option, the maximum altitude for the analysis should not exceed 300,000 feet. An error may occur if the freestream density exceeds the lower bound of the gas table.

A new option was added to the boundary layer (BL) transition setup. This option allows the user to specify whether to have the program check BL transition propagation of each streamline. The program will keep track of the forward propagation of BL transition for each streamline at each time slice and make sure once transition occurs, the BL stays transitional until fully turbulent. A new BL transition method, called the Hypersonic Transition Prediction Module (HTPM) developed by Rockwell International Science Center, was incorporated to the Heat-TK. It is a higher fidelity BL transition method

developed mainly for analysis of hypersonic slender body type vehicle with relatively small angle-of-attack. The range of applicability for the module is the following:

- For smooth bodies with angle of attack  $< 20$  degrees,
- For local Mach number between 5 and 15
- For wall temperature ratio:  $0.2 \leq T_{wall}/T_{adiabatic\ wall} \leq 1$

Required inputs for the HTPM BL transition option are:

- the critical amplification rate, which is inputted via the value at onset
  - For noisy conditions, the critical amplification rate of 5 may be used
  - For quiet conditions, the critical amplification rate of 12 may be used
- The user is also required to specify a multiplication factor (mf) to be applied to transition-onset Reynolds number to obtain the Reynolds number for fully turbulent flow.

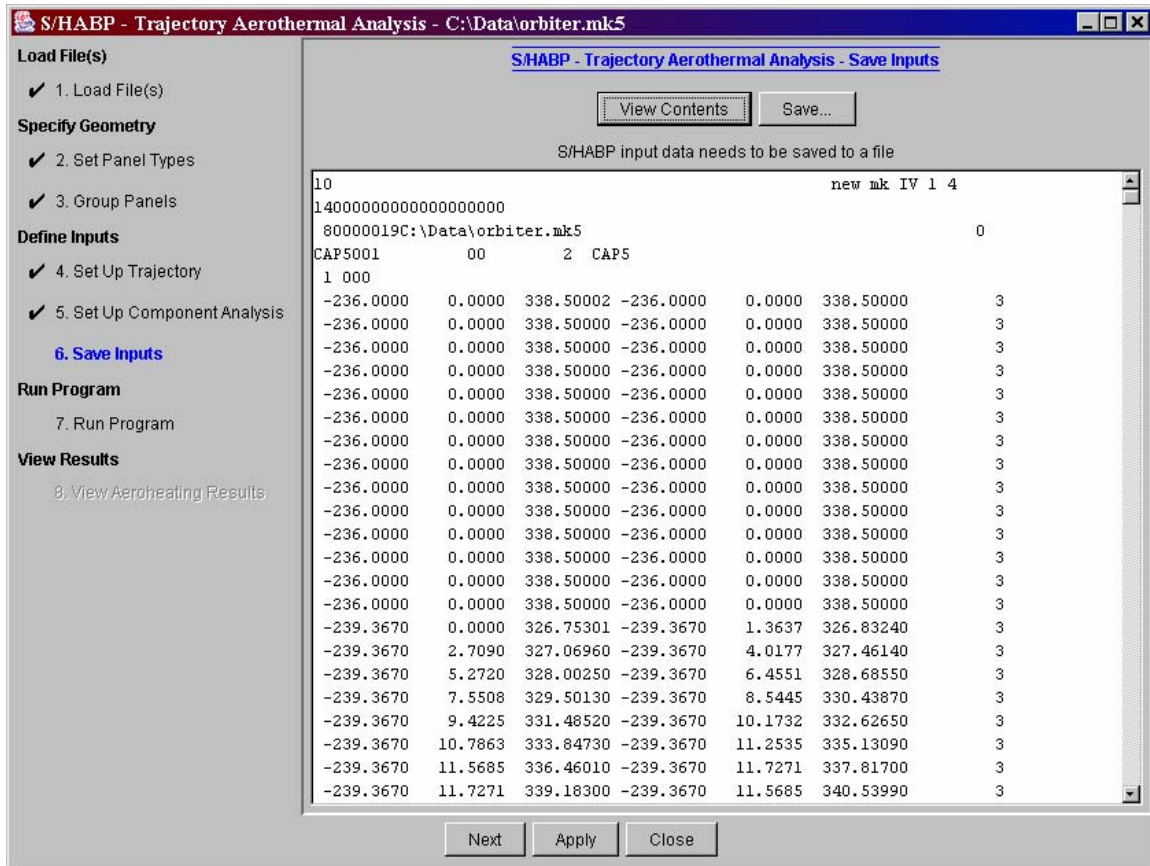
$$Re_{Fully\ Turbulent} = mf * Re_{BL\ Transition\ Onset}$$

It should be noted that in order to use the 2D and/or 3D viewer, the user will need to turn on the print option for the interpolated element.

## 7.6 Save Inputs

Step 6 in the trajectory aeroheating process is used to save the input settings to a file. Figure 45 shows a sample window for the “Save Inputs” step. The “View Contents” button is used to view the contents of the file to be saved – after clicking on this button, the contents appear in the text area. The “Save” button is used to save the contents to a specified file.

**Note:** the values for all the input settings in the trajectory aeroheating process can revert to the last saved values by going back to the “Load Files” step and clicking on the “Load Files” button.

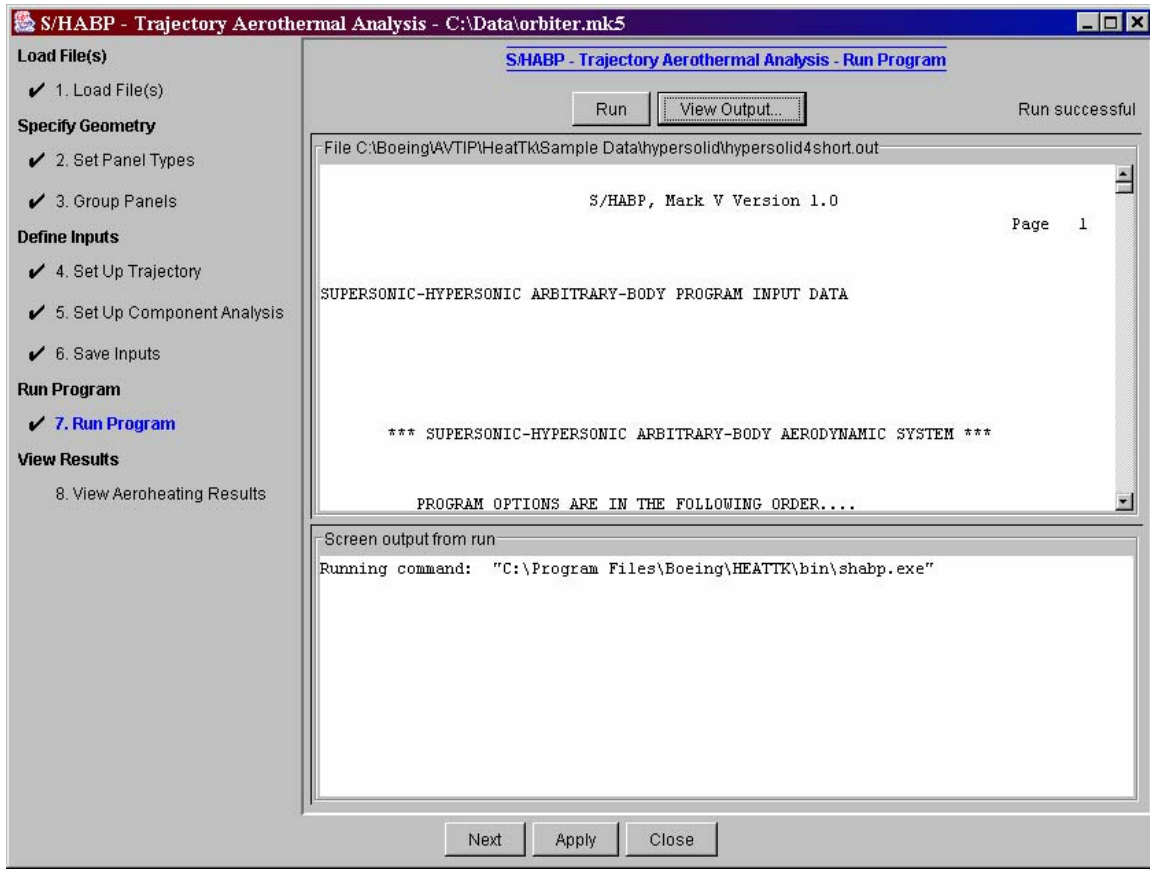


### Figure 45 Trajectory Aeroheating - Save Inputs

## 7.7 Run Program

Step 7 in the trajectory aeroheating process is used to run S/HABP to perform the specified analysis. Figure 46 shows a sample window for the “Run Program” step, as it appears after running the program. The “Run” button is used to run the program. An output file from running the program can be viewed by clicking on the “View Output” button and selecting a file. The contents of the selected file are displayed in the upper text area (unless the file is too large, in which case a separate window is used). The screen output from running the program is provided in the lower text area.

Running S/HABP produces an output file with the name specified in the “Load Files” step. Graphical results can be viewed in the succeeding step.



**Figure 46 Trajectory Aeroheating - Run Program**

## 7.8 View Aeroheating Results

The final step in the process is used to view graphical results from a trajectory aeroheating analysis. The results can be viewed via 2D plots or 3D surface map, or written to a file for viewing with Tecplot®. The following sections outline these capabilities.

### 7.8.1 Two-D Element Plots

Two-D plots of the trajectory aeroheating results for selected vehicle geometry elements are accessed by clicking on the “Two-D Element Plots” tab. Figure 47 shows a sample window for producing 2D element plots. A plot is generated after selecting a wall temperature value, geometry component, what to plot, and one or more geometry elements. The “Plot” option specifies what to plot. If “Custom” is selected, then the x- and y-axis parameters can be specified. If one of the other options is selected (e.g., “Heat Flux vs. Time”), the x- and y-axis parameters are set to correspond to the selection. Clicking on the “Plot” button brings up a separate window with the specified 2D plot. Additional information about the 2D plotter is available in Section 1.2.3.

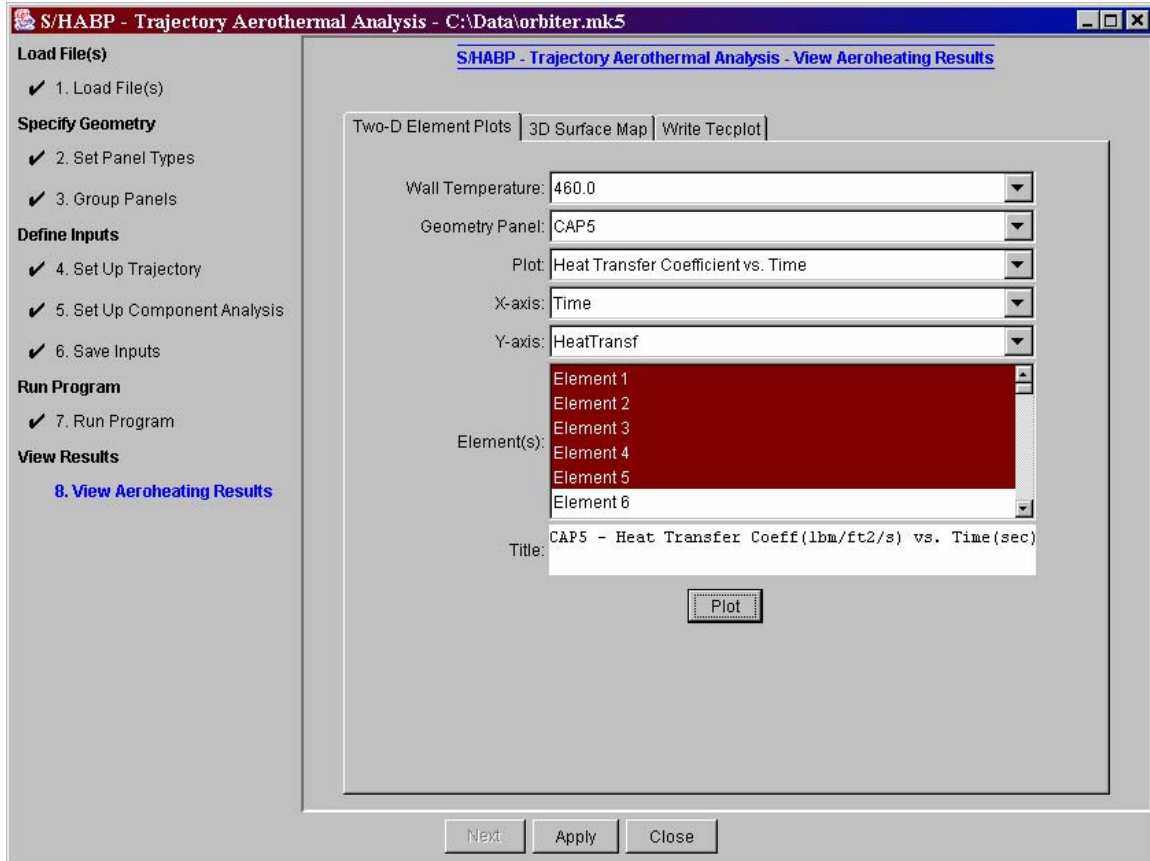


Figure 47 Trajectory Aeroheating - 2D Plots

### 7.8.2 3D Surface Map

Three-D surface plots of the trajectory aeroheating results for selected time slices are accessed by clicking on the “3D Surface Map” tab. Figure 48 shows a sample window for producing 3D surface map plots. A 3D surface view is specified by selecting a wall temperature value, distribution values to display on the vehicle surface, and a single time slice (or maximum over all time slices). The minimum and maximum values of the selected distribution are provided – the default is to generate a 3D view using this range of values. The minimum or maximum value to display can be adjusted. Clicking on the “View” button brings up a separate browser window with the requested 3D view. Additional information about the controls appearing on the right half of the 3D surface map window and about the 3D viewer is available in Section 1.2.4.



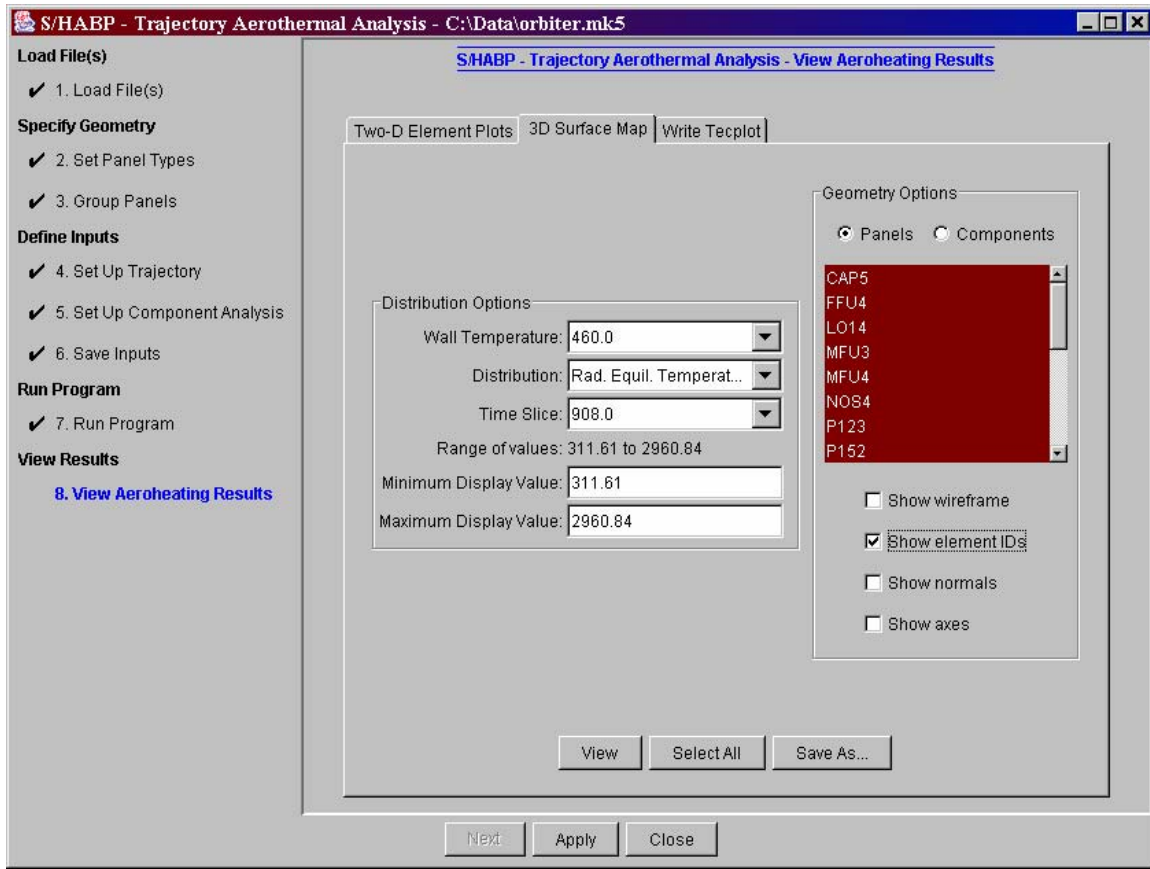
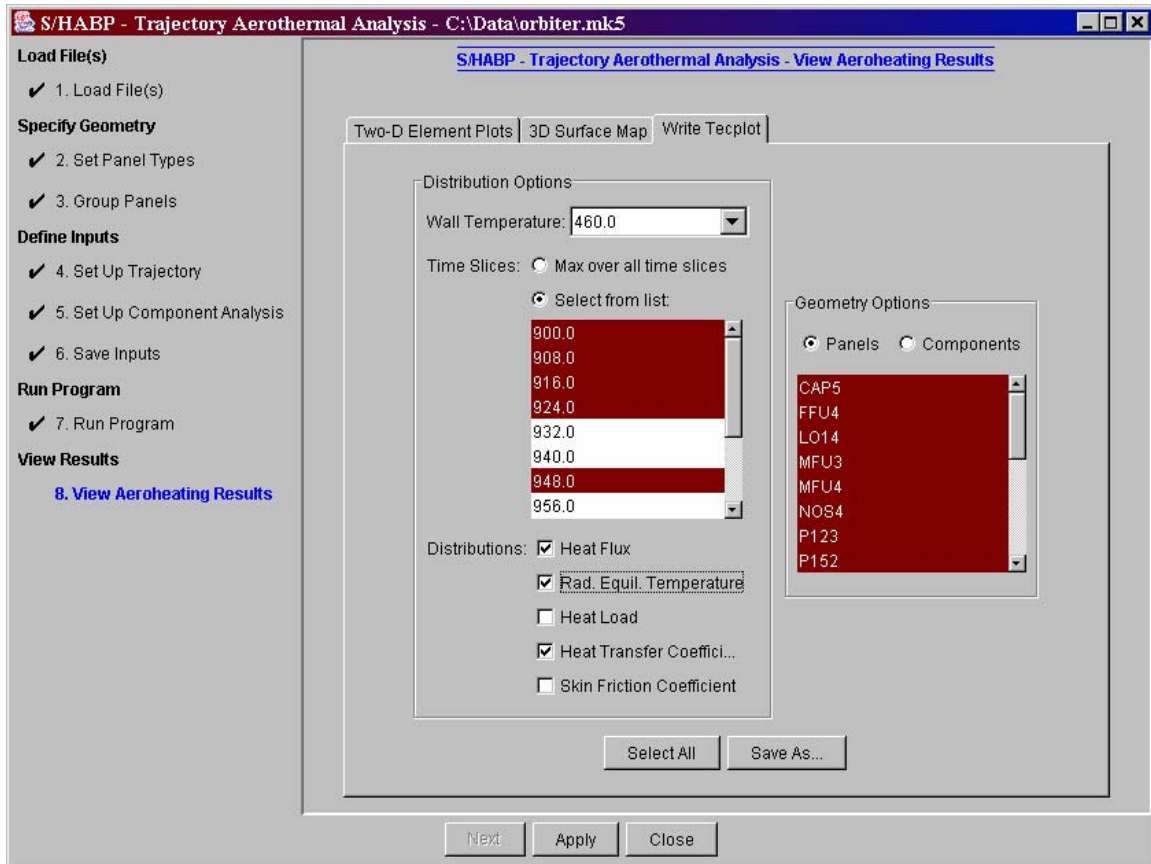


Figure 48 Trajectory Aeroheating - 3D Surface Map

### 7.8.3 Write Tecplot

Tecplot-formatted files containing trajectory aeroheating results are generated by clicking on the “Write Tecplot” tab. Figure 49 shows a sample window for producing Tecplot files. A data set is specified by selecting a wall temperature value, time slice(s) (or maximum over all time slices), distribution(s), and geometry panel(s)/component(s). The specified data is written to a Tecplot file by clicking on the “Save As...” button. The output data will be written in the Tecplot finite-element with FEPOINT file format. The solution obtained for the element centroids is distributed to the element nodes.

**Note:** if “max over all time slices” or a single time slice is selected, the resulting Tecplot file contains one zone per geometry panel (or component). If multiple time slices are selected, the Tecplot file contains one zone per time slice.



**Figure 49 Trajectory Aeroheating - Write Tecplot**

## 8 TPS Sizing Analysis (Chap)

HEAT-TK can be used to perform Thermal Protection System (TPS) sizing analysis on the surfaces of an aerospace vehicle configuration. This TPS sizing analysis capability is accessed from the main window by clicking on the “TPS Sizing” button. A trajectory aeroheating analysis must be performed prior to doing TPS sizing analysis; see Section 7 for documentation related to trajectory aeroheating. The main program that performs the TPS sizing computations is named “Chap”.

There are five steps in the TPS Sizing process: “Load Files”, “Run Model Selection”, “Specify Elements”, “Run Pre-Sizing”, and “Run TPS Sizing”, documented in the following sections.

### 8.1 Load Files

The first step in the TPS sizing process is to load files for analysis. Figure 50 shows a sample window for the “Load Files” step.

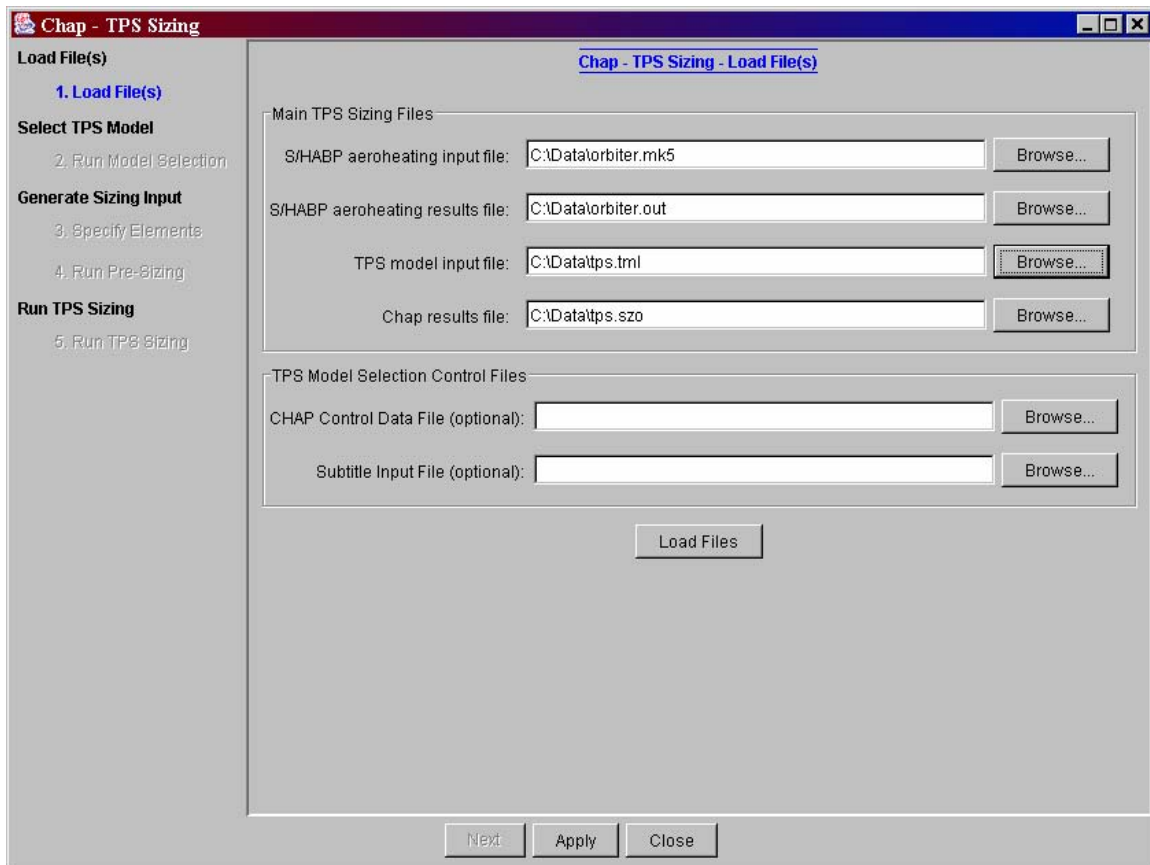


Figure 50 TPS Sizing Load Files

The following files are specified in this window:

- ❑ **S/HABP aeroheating input file** – an S/HABP “.mk5” file used in a successful trajectory aeroheating analysis
- ❑ **S/HABP aeroheating output file** – a “.out” file containing results for a trajectory aeroheating analysis, corresponding to the specified “.mk5” file
- ❑ **TPS model input file** – a TPS material model “.tml” file containing sizing material properties
- ❑ **Chap results file** – the TPS sizing analysis output file to be generated, which will contain the results (a “.szo” file)
- ❑ **Chap control data file [optional]** – a “.tpl” file containing global control parameters for Chap. The control parameters specified in this file will be applied to every Chap analysis cases within the Chap input deck.
- ❑ **Subtitle input file [optional]** – a “.tsb” file containing subtitle information for each surface element. User can use this optional input file to specify subtitle information for each TPS analysis element. To use this option, the user is required to specify the component number, panel number and element number followed by the subtitle of that element (e.g. *comp#*, *panel#*, *element#*, *subtitle*). One subtitle per element.

After the files have been specified, click the “Load Files” button to load information from these files into the TPS sizing process. If it is successful, then succeeding step(s) in the process will be accessible.

## 8.2 Run Model Selection

The second step in the TPS sizing process is to run a preprocessor program (selecttps1). The preprocessor will automatically select the type of TPS for each vehicle surface element. The selection is based on the surface equilibrium temperature generated from the aeroheating analysis for each element. This will be discussed in more detail in section 8.6.1. The user can also manually select the appropriate TPS to use after the “Run Model Selection” step has been completed or in case the preprocessor cannot locate the TPS model with enough temperature range. This will be discussed in Section 8.3. If this occurs, the user may have to introduce a new TPS model that can handle the temperature range required.

The procedures for setting up the TPS model are specified in the CHAP manual under Section 1.2.1.1 “Material Data.” Figure 51 shows an existing model already set up in the TPS model input file. The model is setup for the CRI TPS. The first line specifies the TPS model name, followed by the temperature range which will be used for TPS model selection process. The second line specifies the name of the TPS layer material, the thickness of the material and the number of nodes for this layer. If the user chooses to size this TPS material, several options are available to the user. The first parameter after the number of nodes input is the flag used to turn on the option for TPS sizing. For example, Line 14 specified the third TPS layer and also specified that AFRSI are to be sized. No sizing is the default. The second parameter used to indicate which node

temperature will be used to size the TPS material. The parameters from this point on are active only if the sizing flag has been turned on. For example, Line 30 in specifies the last layer of the TPS and also specifies that the AFRSI material is to be sized to the first node temperature of the structure. The one follows the sizing flag. The next parameter allows the user to specify the limiting TPS sizing temperature. The default limit is 350 degree Fahrenheit. Note that the program is not setup to size multiple layers.

Six additional parameters following the limiting sizing temperature input are available to provide user some control over the sizing process. The sizing process consists of two basic steps. The first step allows the program to fast home in on an approximate TPS sizing thickness that will satisfy the specified TPS sizing temperature limit. The second step refines the sizing thickness by using the Newtonian iteration method. The next three parameters are available to the user to control the first step process. The next two parameters control the range of sizing thickness (with default values of 0.5 to 25.0) and the third parameter specifies the step size (with default value of 0.2). The last three parameters control the Newtonian iteration process. These parameters are the maximum number of iterations (with the default of 20 iterations), the iteration step size (with the default of 0.0001), and the sizing tolerance (with the default of 1.0e-6) (e.g. AFRSI,0.138,10,1,,,5,25.0,0.2,20,0.0001,1.0e-6). Finally, the TPS model ends with an “end” in the last line in the model.

Figure 52 shows a sample window for the “Run Model Selection” step, as it appears after running the program. The “Run” button is used to run the program. An output file from running the program can be viewed by clicking on the “View Output” button and selecting a file. The contents of the selected file are displayed in the upper text area (unless the file is too large, in which case a separate window is used). The screen output from running the program is provided in the lower text area.

Running the selecttps1 preprocessor program in this step produces several files:

- A “.psz” file containing TPS models selected for each surface element,
- A “.htb” file containing heating data in a time history table format, and
- A “.err” file that lists any surface locations that failed to find TPS models

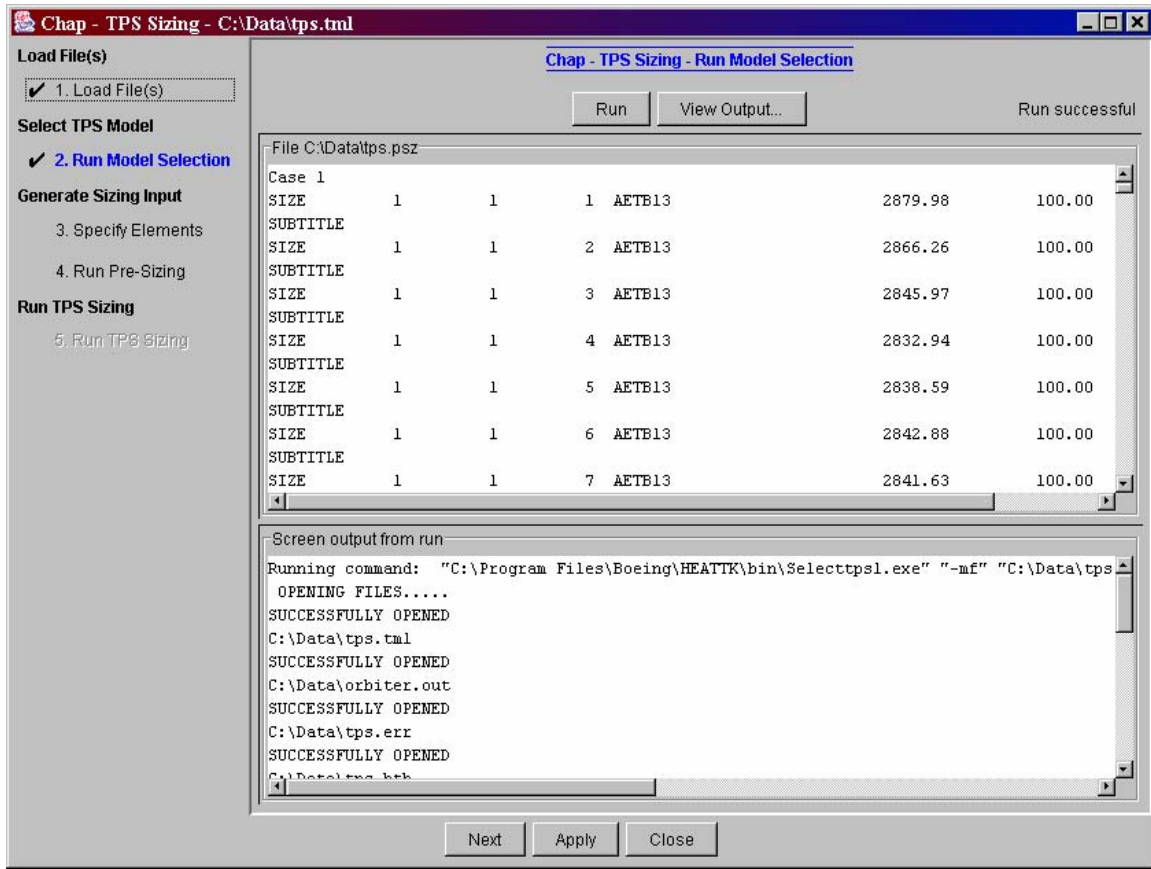
The prefix for each of these files is the same as the prefix of the TPS model input file specified previously.

```

CRI          1501.0    2200.0
C9,0.02,10
VIRGIN,125,TABLE,TABLE,TABLE
0,500,750,1000,1250,1500,2000
.00006,.00014,.00018,.00022,.00027,.00032,.00048
0,500,750,1000,1250,1500,2000
.177,.244,.27,.277,.269,.282,.307
0,500,750,1000,1250,1500,2000
0.87,0.81,0.71,0.66,0.59,0.54,0.45
NEXTEL , .05 , 10.
VIRGIN , 60. , 6.E-4, TABLE , .8
32. , 392. , 752. , 1472. , 1832.
.1 , .22 , .275 , .27 , .39
AFRSI,0.138,10,1
VIRGIN,8.1,TABLE,TABLE
0,500,1000,1250,1500,1750
6.34E-05,1.37E-05,2.20E-04,2.66E-04,3.17E-04,4.0E-
04
0,500,1000,1250,1500,1750
.177,.244,.277,.269,.282,.295
NEXTEL , .03 , 10.
VIRGIN , 60. , 6.E-4, TABLE , .8
32. , 392. , 752. , 1472. , 1832.
.1 , .22 , .275 , .27 , .39
RTV,0.012,10
VIRGIN,88,TABLE,TABLE
0,200,400,560
0.0008172,0.0006672,0.0005436,0.0004872
0,200,400,560
0.265,0.300,0.340,0.340
6AL4V,0.05,10,0,1,350

```

Figure 51 A Simple TPS Model Generated for the CRI TPS



**Figure 52 TPS Sizing - Run Model Selection**

### 8.3 Specify Elements

The third step in the TPS sizing process is to specify geometry panels or individual geometry elements to use in the sizing analysis. Performing a sizing analysis on an entire vehicle normally takes an extended amount of time to run and should be done only when necessary. Figure 53 shows a sample window for the “Specify Elements” step.

Information about which geometry elements and associated models to use in a sizing analysis is stored in a “.pem” file. This file contains one line for each element to include, with each line consisting of four parameters – a geometry component number, panel number, element number, and material model identifier. A “.pem” file can be specified at the top of the “Specify Elements” window and loaded via the “Load” button. The data from a loaded file is depicted in the table at the bottom of the window, in a simpler format.

Geometry elements for selected panels and an associated model can be added to this table via the middle section of the window. This is done by selecting a geometry panel from the list, one or more elements, a model, and clicking on the “Add to Table” button. A 3D

view of panels/elements for the vehicle geometry can be accessed from the “View Panels” button – see Section 1.2.4 for additional information about the 3D viewer. Geometry elements can be removed from the table by selecting one or more rows of the table and clicking on the “Remove” button.

After the table is updated, the results can be saved to a “.pem” file by clicking on the “Save” button.

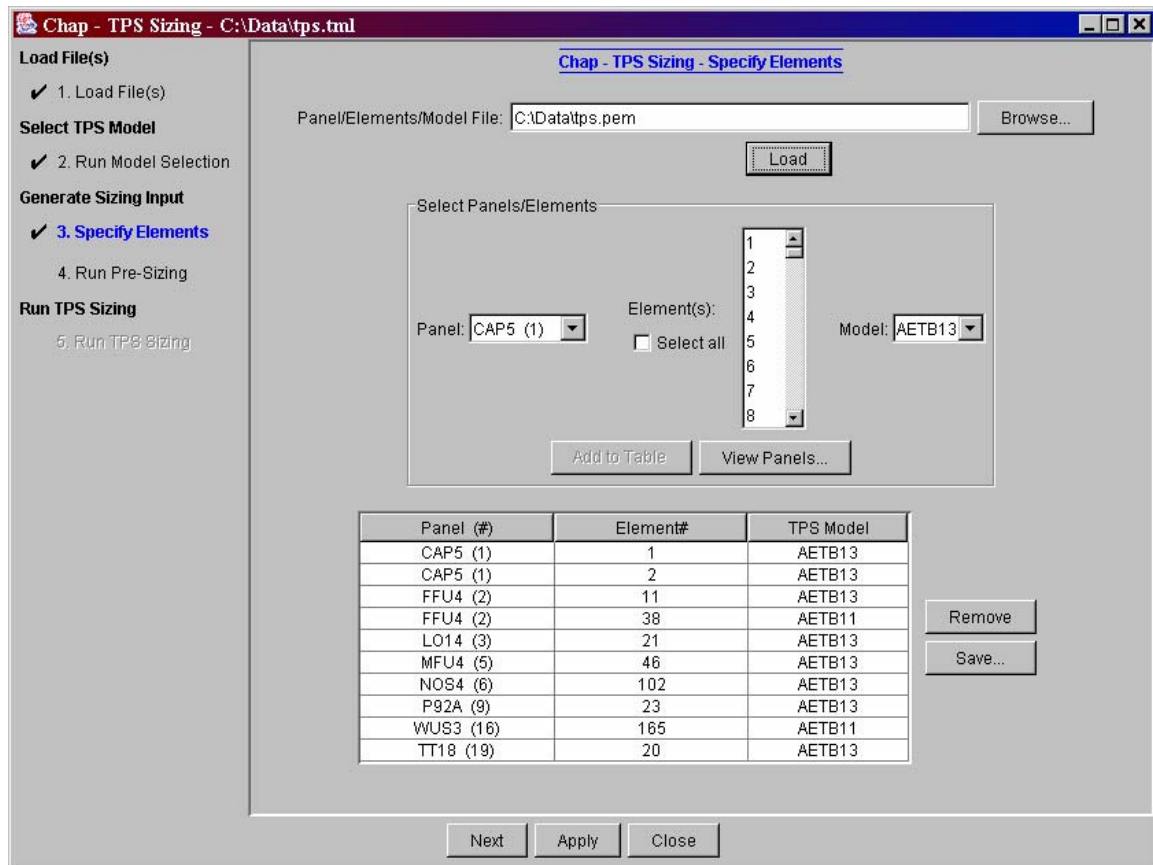


Figure 53 TPS Sizing - Specify Elements

## 8.4 Run Pre-Sizing

The fourth step in the TPS sizing process is to run a second preprocessor program (prechap2) that automatically sets up an input file for Chap, based on data generated in the previous steps. Figure 54 shows a sample window for the “Run Pre-Sizing” step, as it appears after running the program. The “Run” button is used to run the program. An output file from running the program can be viewed by clicking on the “View Output” button and selecting a file (unless the file is too large, in which case a separate window is used). The contents of the selected file are displayed in the upper text area. The screen output from running the program is provided in the lower text area.



Running the prechap2 preprocessor program in this step produces a “.szi” file, which is used as input to the next step. The prefix for this file is the same as the prefix of the TPS model input file specified previously.

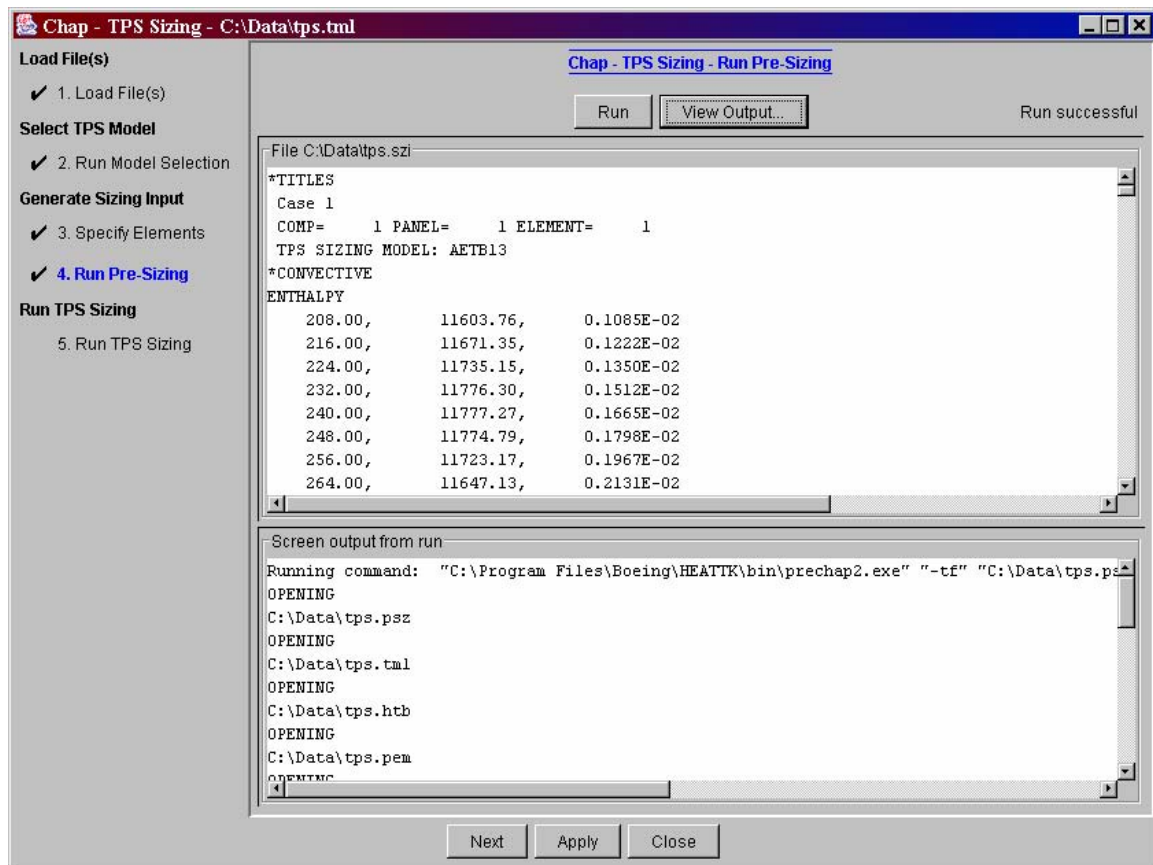


Figure 54 TPS Sizing - Run Pre-Sizing

## 8.5 Run TPS Sizing

The final step in the TPS sizing process is to run Chap, the TPS sizing program. Figure 55 shows a sample window for the “Run TPS Sizing” step, as it appears after running the program. The “Run” button is used to run the program. An output file from running the program can be viewed by clicking on the “View Output” button and selecting a file. The contents of the selected file are displayed in the upper text area (unless the file is too large, in which case a separate window is used). The screen output from running the program is provided in the lower text area.

Running the TPS sizing program in this step produces the following files:

- ❑ A “.szo” file containing summary TPS sizing information for the specified surface elements, and
- ❑ A “.szf” file containing detailed information from the TPS sizing program

The prefix for each of these files is the same as the prefix of the TPS model input file specified previously.

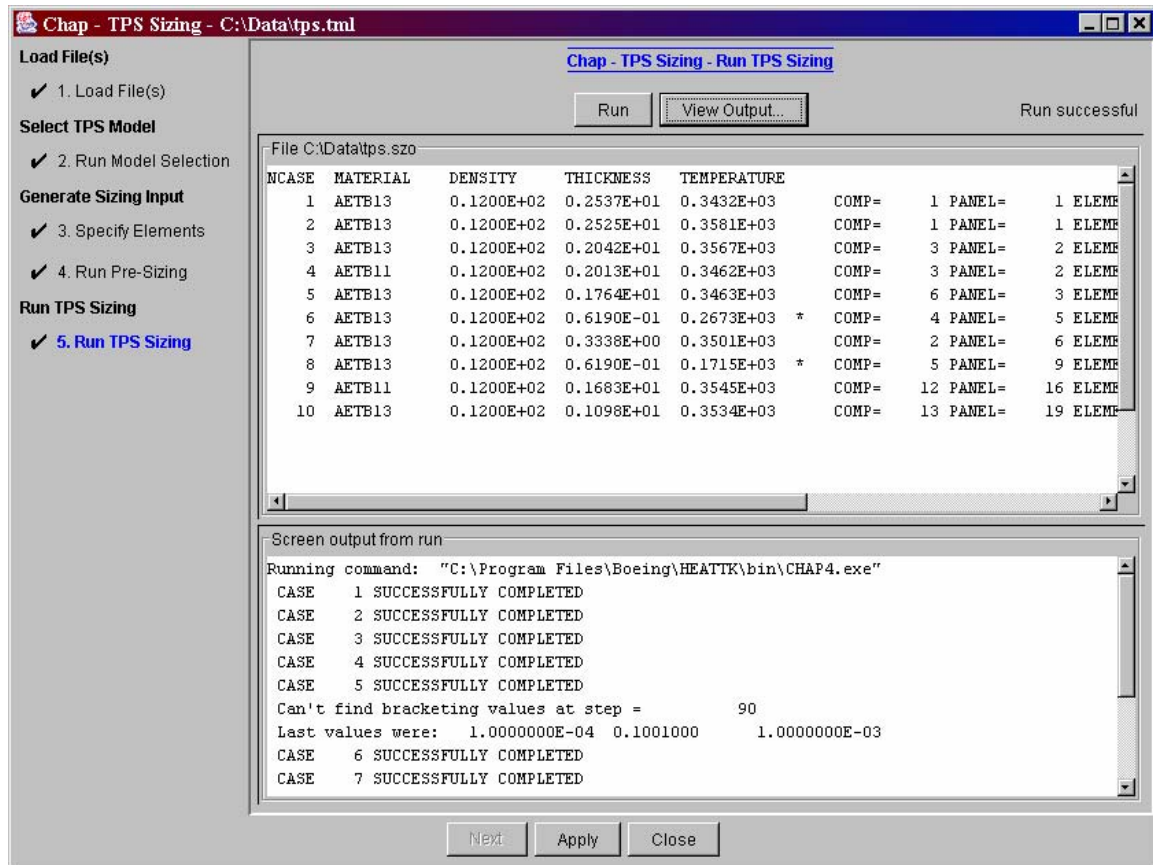


Figure 55 Run TPS Sizing

## 8.6 TPS Sizing Programs

This section provides some additional documentation for the TPS sizing programs. Although this information is not typically needed in using the HEAT-TK GUI, it is included here for completeness.

### 8.6.1 Selecttps1

The selecttps1 program automatically selects, based on the surface equilibrium temperature, the type of TPS for the vehicle surface element. The program reads from a HEAT-TK trajectory aeroheating results file the following information:

1. Run Title
2. Panel ID
3. Element ID
4. Wall Temperature
5. Time

6. Heat transfer coefficient
7. Adiabatic wall enthalpy
8. Equilibrium Temperature

The program calculates the maximum surface panels and elements and the total number of time cuts based on the information provided in the trajectory aeroheating output file. The program dynamically allocates the required storage for the program variables. If the program failed to extract the required information from the output file, it will default to the following dimensions:

Max. panels = 30  
 Max. elements = 1000  
 Max. time cuts = 200  
 Max. wall temperatures = 6

The user can also manually input the variable dimension using the command line input. For example,

```
selecttps1 -c 100 -p 50 -e 500 -w 4
```

where

-c (number of time cuts)  
 -p (number of panels)  
 -e (number of elements)  
 -w (number of wall temperature)

The program will read from the following files:

- ❑ TPSMODEL.TML – the TPS model file
- ❑ SHABP.OUT – trajectory aeroheating results file
- ❑ TPSSIZE.TPL – optional control input data for CHAP
- ❑ SUBTITL.TXT – optional sub title input file for each surface element. Note that this activates only when a user specifies the -sf command line option parameter. If no file name specified then the program will use the default file name SUBTITL.TXT.

Output files:

- ❑ MATSEL.ERR – log file that lists surface locations that fail to find TPS models
- ❑ HEATDATA.HTB – HEAT-TK heating data reformatted into the time history table format for the prechap2 program
- ❑ TPSSIZE.PSZ – TPS model selections output file for prechap2 program

Note that the above file names are the default file names for input/output to Selecttps1. The user can manually specify the file by using the command line input options:

-sf (optional sub title file)  
 -mf (TPS material model file)  
 -lf (log file that outputs the list of failed TPS model selections)  
 -hf (the heat table output)  
 -vf (trajectory aeroheating results file)

- gf (optional Chap control input)
- cf (TPS selection output)

**Note:** the HEAT-TK GUI uses the command-line options to generate a set of files having the same prefix.

### 8.6.2 Prechap2

The prechap2 program will automatically setup the Chap input deck from the output of the selecttps1 program. The required input files:

- ❑ TPSSIZE.PSZ – the output TPS selection file from selecttps1
- ❑ TPSMODEL.TML – the TPS material model file
- ❑ HEATDATA.HTB – the heating time history table file
- ❑ TPSSEL.INP – a file specifying what geometry elements and associated models are to be used in the TPS sizing analysis. This file consists of lines each containing a geometry component number, panel number, element number, and material model identifier.

Output:

- ❑ dat.sample – Chap input deck

The file names can be specified using the command line input options:

- tf (the output TPS selection file from selecttps1)
- mf (the TPS material model file)
- hf (the heating time history table file)
- pf (the geometry element/model file)
- cf (Chap input deck)

**Note:** the HEAT-TK GUI uses the command-line options to generate files having the same prefix.

### 8.6.3 Chap

The chap program performs TPS sizing analysis based on the output from prechap2. The required input file is dat.sample, created by prechap2. It produces the following output files:

- ❑ size.sample – summary results of TPS sizing analysis
- ❑ out.sample – detailed results of TPS sizing analysis

**Note:** the HEAT-TK GUI renames the output files from chap so they have the same prefix.

## 9 References

1. Burns, Keith A., et. al., “Viscous Effects on Complex Configurations, Software User’s Manual”, WL-TR-95-3060, Aug. 1995.
2. Deters, K.J., *QUADSTREAM User’s and Reference Manual*, WL-TR-92-3071, May 1992.
3. *NASP Phase II-D Final Report*, Section 2.5.4.1, Report Number X30NP94006A, Dec. 1994.
4. *The Convective Heating and Ablation Program (CHAP), Volume 1, Theoretical Manual*, Boeing Document No. D180-28494-1, March 1992.

## Appendix A

### Subsonic – Supersonic AEROdynamics (SSAERO)

To calculate aerodynamic coefficients of the various components of a flight vehicle, SSAERO uses a variety of methods. Typically, a flight vehicle is divided into a body component and lifting surface components such as wings, tails, fins, etc. Since the geometry discretization (grid) requirements are different from that of S/HABP, the grid generation has to be done external to HEAT-TK.

#### **Geometry Discretization**

The body geometry is discretized as a structured grid of cross sections at constant axial stations with equal number of points at each cross-section. It must have at least two cross sections with at least three points per section. The lifting surfaces are modeled as airfoil sections with constant number of points per section. The ordering of points is shown in . A pseudo code in Figure 56 shows the format of the geometry file. The name of this file should have the extension '.geo'. The name of parts in the geometry file should correspond to one the keywords given in . The flags that mark the various segments of a part are shown in .

Table 3 Ordering of Points in Geometry File

Part	1 <sup>st</sup> Section	Last Section	Direction of Points
Body	Nose	Aft Base	Bottom to Top
Tail/Fin	Root	Tip	LE-inboard-TE-outboard-LE
Centerline Tail	Root	Tip	TE-outboard-LE
Wing	Root	Tip	LE-upper-TE-lower-LE

```
Do n = 1,Nparts
  write (unit, '(a4)') PartName
  write (unit,*)
  do j = 1,NJ(n)
    do i = 1,NI(n)
      write (unit,'(3f10.4,i1)') x(i,j), y(i,j), z(i,j), Flag
    end do
  end do
end do
```

Figure 56 Pseudo-code Showing Geometry Format

**Table 4 Keywords to Be Used to Name Parts**

<b>Keyword</b>	<b>Description</b>
BODY	The fuselage
FINS	Any aerosurface
TAIL	Vertical aerosurface on the centerline
CFIN	A fin which can be deflected to trim
WING	A fin which is the primary lifting surface

**Table 5 Flags to Make Various Segments of a Part**

<b>Flag</b>	<b>Description</b>
2	Beginning of a part
1	Beginning of a section within a part
3	End of a part

## Aerodynamic Analysis Methods

Methodologies used for building the aerodynamic coefficients of the body and the lifting surface geometries are given in and , respectively.

**Table 6 Methods Used for Aerodynamic Analysis of the Body**

<b>Aerodynamic Coefficients</b>	<b>Method</b>	<b>Description</b>
Linear Normal Force, Pitching Moment, Side Force, and Yawing Moment	Slender Body Theory (SBT)	The governing potential flow equation is reduced to two-dimensional Laplace equation and solved using a boundary element technique where the body cross sections are divided into short segments along which constant strength singularity lines are distributed. SBT predicts the loading on a body independent of the freestream Mach number. Consequently, it is used at all Mach numbers.
Non-Linear Normal Force, Pitching Moment, Side Force, and Yawing Moment	Viscous Crossflow Theory (VCT)	VCT calculates the non-linear contribution to the normal force. It has been validated for angles-of-attack up to 90°. The non-linear normal force is a function of crossflow drag coefficient, fineness ratio, body planform and angle-of-attack. The crossflow drag coefficient for circular cross sections is corrected for non-circular sections using a ratio of 2-D Newtonian drag for circular cross sections.
Wing Body Interference Factor	Upwash Theory	Upwash theory provides an accurate method to calculate the change in lift on a wing due to the upwash theory from the body. SBT predicts the upwash from the body panel source strengths.
Potential Axial Force	SBT	The potential axial force is determined based on slender body theory and viscous crossflow theory. The potential force vector is assumed to lie midway between the lift and the normal force vectors.
Viscous Axial Force	Van Driest	Viscous axial force is calculated using Van Driest's method throughout the Mach number range. The Reynolds number is calculated as a function of Mach number and altitude. The wetted area and running length are determined from the geometry.
Form Axial Force	Emperical Correction Based on methods from Blakediee, Johnson, and Skavdahl	The body form axial force is determined as a function of skin friction and body fineness ratio.
Base Drag	Collection of Empirical Data from Payne, Hartly, and Taylor	Base drag is calculated for geometries that have aft panels with slopes greater than 85°.
Wave Drag	Second Order Shock Expansion (SOSE)	SOSE is based on the local surface inclination angle and assumes a Prandtl-Meyer expansion downstream from the nose. The method uses an effective body of revolution, which is determined from the input geometry.



**Table 7 Methods Used for Aerodynamic Analysis of Lifting Surfaces**

<b>Aerodynamic Coefficients</b>	<b>Method</b>	<b>Description</b>
Lift and Pitching Moment	Vortex Lattice Method, Linear Theory and Slender Wing Theory	A vortex lattice method (WINGCP3) is used to predict subsonic lift and pitching moment for all lifting surfaces. This method accounts for planform and camber effects. Supersonic aerodynamics is calculated using methods in USAF Stability & Control DATCOM handbooks. These methods account for taper ratio, aspect ratio and sweep angle.
Induced Drag	$C_D = k C_L^2$	This is a standard method for calculating induced drag for lifting surfaces. The subsonic planform correction factor takes into account aspect ratio, taper ratio and other planform parameters. The supersonic planform correction factor includes Mach number also.
Viscous Drag	Van Driest	See description for body viscous axial force
Form Drag	Empirical Correlation from Shevllle	Subsonic form drag is calculated up to the crest critical Mach number. The form drag is a function of Mach number, thickness ratio and leading edge sweep.
Wave Drag	Linear Theory	Wave drag is calculated for both subsonic and supersonic leading edge wings. Both methods account for Mach number, thickness ratio and airfoil form factor. In addition, the supersonic leading edge method accounts for leading edge sweep.

## **Appendix B**

### **Mission Analysis Program (MAP)**

The Mission Analysis Program (MAP) computes the flight performance of an aircraft when operated over a specified mission or sequence of operations. Several different types of operations (legs) may be computed and combined according to user specified rules governing the overall type of mission (range, radius, time on station, etc.). In addition, sub-sets of legs may be constrained to cover a specified time or distance, or match the distance traveled in a second sub-set of legs.

The types of operations which may be computed include: Fixed power maneuvers such as climb and/or accelerate, descend and/or decelerate, and trajectory legs; Part power maneuvers such as maximum range (cruise), maximum endurance (loiter), refueling, and constant thrust to weight; Miscellaneous operations such as warm-up, take-off, combat, fuel reserve, and store release. Operations are computed primarily in the vertical plane of motion assuming a point mass vehicle and a spherical non-rotating earth. Six different atmosphere models (including U.S. Standard) are available within the program with provisions for defining non-standard atmospheres. Acceleration due to gravity is determined from the inverse square law.

Fixed power maneuvers may be computed along a specified mach/altitude trajectory or along minimum time or minimum fuel trajectories. The energy method is used to define these “optimum” flight paths. Maximum range descents may be approximated by operation at maximum L/D ratio. Fixed power maneuvers may also be computed along a trajectory defined by one of several options for controlling angle of attack. Part power maneuvers may be computed at optimum or fixed mach, and optimum or fixed altitude.

Input to the program includes tables defining aerodynamic and installed propulsion characteristics as functions of mach, altitude, power setting, angle of attack, and configuration. Multiple engine types may be accommodated as well as engine-out operation. Thrust deflection angles may be varied from leg-to-leg but not within a given leg. Aircraft weights at the start and end of the mission are input as well as the weight of fuel available and stores or external tanks released.

Outputs from the program include detailed printouts of the time history of each leg as well as a summary print of the complete mission. A time history of user specified parameters may be output to a file for one or more mission legs. Individual legs or sequences of legs may be computed and stored internally or on files for use in later cases or on future computer runs. Multiple cases may be computed on any given computer run. Payload-weight trades or other performance trades may be generated quickly using this multiple case feature.

The program is written in Fortran 77 and currently runs on a Silicon Graphics workstation running the IRIX 6.5 Unix operating system. A subset of MAP was compiled, linked, and tested on a PC for this contract.